Introductory Examples

InfoCAD Program System

The description of program functions within this documentation should not be considered a warranty of product features. All warranty and liability claims arising from the use of this documentation are excluded.

InfoGraph® is a registered trademark of InfoGraph GmbH, Aachen, Germany. The manufacturer and product names mentioned below are trademarks of their respective owners.

This documentation is copyright protected. Reproduction, duplication, translation or electronic storage of this document or parts thereof is subject to the written permission of InfoGraph GmbH.

© 2015 InfoGraph GmbH, Aachen, Germany. All rights reserved.

Table of Contents

Introduction to the InfoCAD Program System 2
- InfoCAD Study Version 2
- Installation 2

Using the Program 3
- Program Start 3
- Program Interface 4
- Design Codes 4
- Basic Procedures 5
- Selecting 6
- Entering Coordinates 6

Example 1: Slab with Downstand Beam 7
- Drawing Ground Plans with Model Objects 8
- Specifying Supports and Cross-Sections 10
- Generate the FEM Mesh 11
- Defining Load Cases 11
- Specifying Actions and Design Situations 13
- Performing Calculations 15
- Processing Results 15
- EN 1992-1-1 Checks 16
- Printing 18
- Printing List 19

Example 2: 2D Hall Frame 20
- Task 20
- Specifying the Framework 21
- Defining the Section 22
- Checking the Structure Properties 22
- Defining Load Cases 23
- EN 1993-1-1 Steel Checks 24
- Performing Calculations 26
- Processing Results 26
- Printing 27
- Printing List 28
- InfoGraph Systemviewer 29

Example 3: 3D Building 30
- Task 30
- Drawing the wire frame model 31
- Assigning Properties 33
- Generating an Element Mesh 35
- Defining Load Cases 36
- Describing Actions 38
- Calculation and Results 39
- InfoGraph Systemviewer 40

Example 4: Prestressed Roof Girder 41
- Task 41
- Static System 43
- Loads and actions according to DIN EN 1992-1-1 47
- Calculation and results 49

© InfoGraph GmbH, July 2015

Introduction to the InfoCAD Program System

This documentation is intended to help you get started with editing beam and shell structures and to demonstrate the various options available for displaying the results. You can press the F1 key to access the online help for the function you are currently using in the program. The help system also contains the complete documentation of the program, including explanations of theoretical principles, calculation methods and results.

During installation example files will be saved to the user directory under *Infograph/Samples*. They include projects with beam, cable and shell structures as well as applications relating to structural dynamics and prestressed concrete construction. The examples are provided without results. To view the results, you first need to perform the relevant calculation.

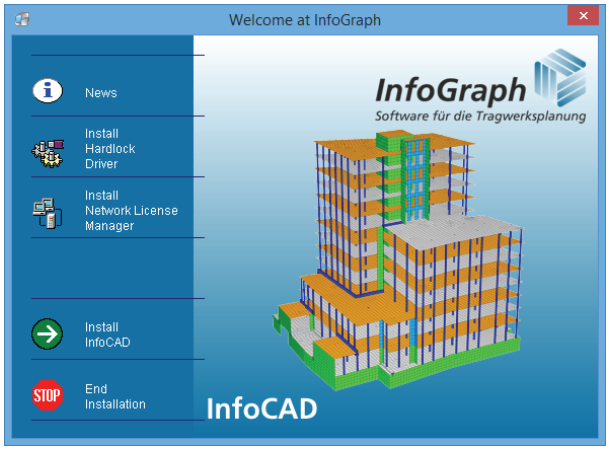
Product news, user tips and updated versions of the program are available at **www.infograph.eu**.

InfoCAD Study Version

The **InfoCAD Study Version** was created as a way to get acquainted with the program and its features by using the examples or your own projects.
The study version provides the same user interface, calculation and checking methods and results output as the full version. The only difference lies in the program capacity, which is limited as follows:

Finite elements:	1000 elements
	1 area section
	1 beam section
Dynamics:	1 eigenmode
Prestressing:	1 tendon
Frameworks:	10 beams
	1 section

Installation



The program is compatible with Windows XP or higher. Administrator privileges are required to perform the installation. Only basic user privileges are needed to use the program.

Windows will automatically start the installation when you insert the CD. Alternatively, you can also start the installation with the *Add or Remove Programs* tool in the Windows *Control Panel* or by executing the setup.exe file on the CD. Updated installation files infocad_xxx.exe and infocad_std_xxx.exe (study version) can be downloaded from the service page of our website.

© InfoGraph GmbH, July 2015

Installation requires the following work steps:

1. Install the software protection (not necessary for the study version)
   Plug the software hardlock into the USB port, insert the program CD and click *Install Hardlock Driver*.

2. Install the program
   Click *Install InfoCAD* in the window shown above and follow the on-screen instructions. The license number you need to enter is located on the back of the CD case (not necessary for the study version).
   All application and help files will be stored in the selected target folder.

The following applications will be installed and can be launched from the Start menu:

1. **InfoCAD** or **InfoCAD Study Version**
   (main program system for static calculations and checks)

2. **InfoGraph Systemviewer**
   (for system visualization and deformation animation)

3. **InfoGraph Crack Width Limitation**
   (stand-alone program)

4. **InfoGraph Lateral Torsional Buckling Check**
   (stand-alone program)

You can remove the program system with the *Add or Remove Programs* tool in the *Control Panel*, which contains the automatic uninstallation program for InfoCAD. Uninstallation will not affect files created or modified by the user.

Using the Program

Program Start

To start the program, go to the *Programs / InfoGraph* folder in the *Start* menu and select **InfoCAD** or **InfoCAD Study Version**. An empty program window will appear.

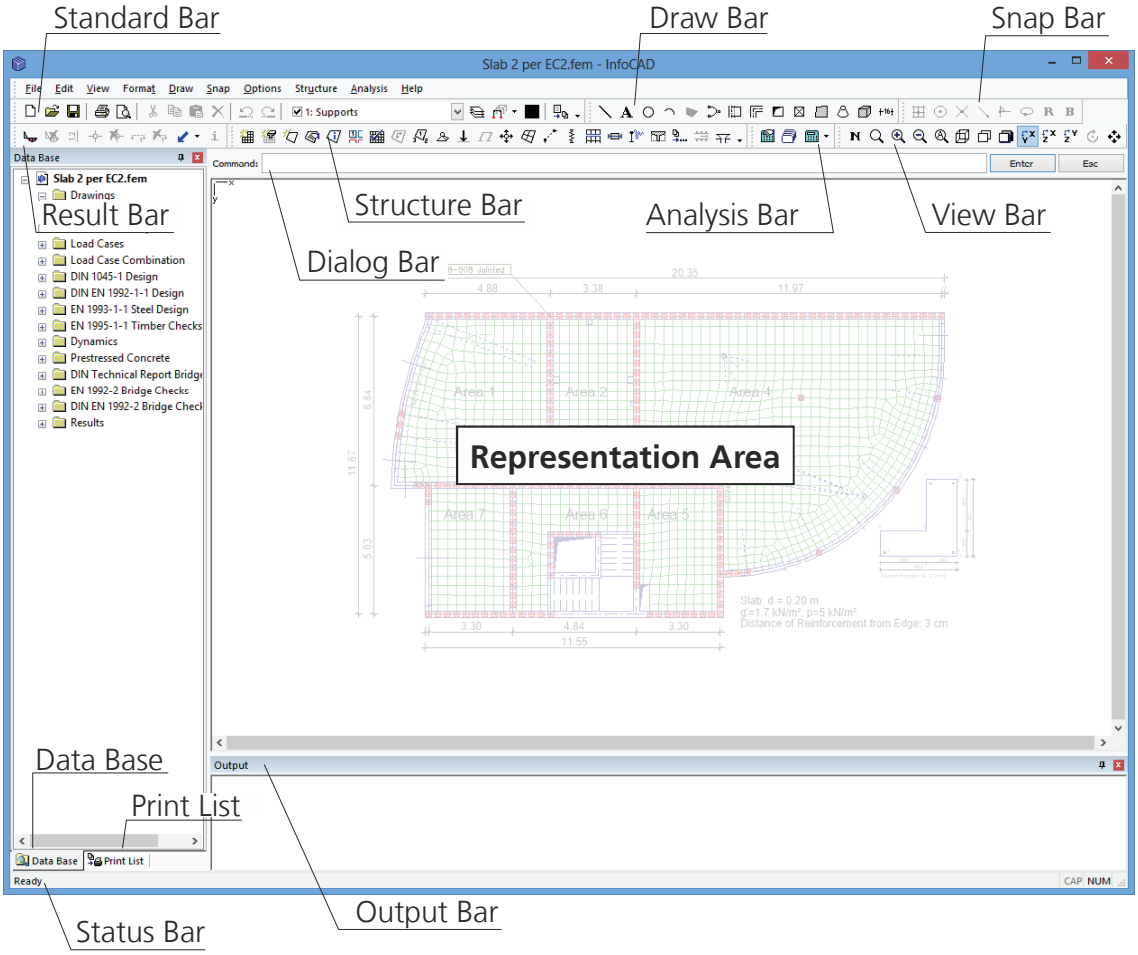
Click the *Open* button to access InfoGraph project files with the following extensions (e.g., example files):

File name	.fem	->	Finite Element Project
	.rsw	->	3D Frame
	.esw	->	2D Frame
	.ros	->	Axisymmetric shell

These file types are registered during installation, which means they can be opened by simply double-clicking the file.

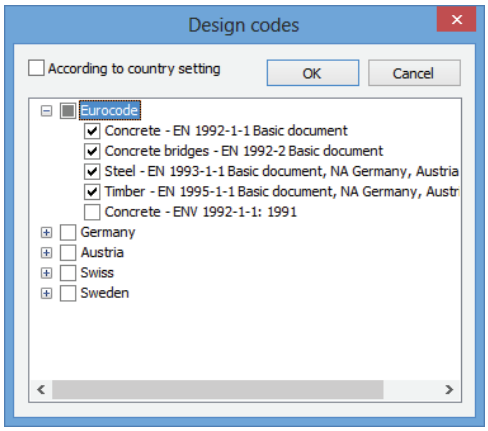
Program Interface

The figure below highlights the most important controls of the **InfoCAD** program interface. For FEM projects, the xy plane appears in the representation area after you select the structure type. Slab and plain stress structures are generally specified in this plane.
Functions can be accessed from a tool bar, a menu item, the dialog bar or the workspace. For graphical functions you normally have to enter additional data, which is then queried in either a separate window or the dialog bar.



Design Codes

The design codes and associated materials available in the program correspond to the regional settings of your computer. If necessary, you can change these settings in the *Options* menu.



© InfoGraph GmbH, July 2015

Basic Procedures

Structures are normally specified and calculated using the following work steps:

1. Describe the problem with the model objects
2. Specify supports and cross-sections
3. Generate an element mesh
4. Define load cases
5. Define the actions and design situations
6. Perform calculations
7. Display or print out the results

Model objects are critically important for system generation

Model objects describe the geometry and properties of individual structural components. They form the basis for the program-controlled generation of finite element meshes and are accounted for the automatic mesh generators *form-sensitive* and *grid-shaped*.

The following model objects are used:

Edge Edges define the borders and the axes of structural components. Depending on their properties, they can also define a line support, a continuous beam or free beams (e.g. columns). Edges are normally used to describe the whole structure in a wire frame model.

Wall Like edges, walls define the border of a structural component as well as a line support. The geometry of a component is determined by the wall axes. Walls are specifically suited for creating pure slab systems.

Hole Holes define openings in the element mesh. They can also define line supports and continuous beams according to their properties.

Column Columns define a point support or, alternatively, a column head condensation in the element mesh.

Face Faces describe areas that are automatically meshed by the mesh generator. Edges, walls and holes form the outer border of these meshes. All model objects inside the face are taken into consideration during mesh generation. The face can be assigned various properties for the FEM mesh.

Cone The *Cone* model object describes a truncated cone segment for subsequent meshing. You can use the shortcut menu to determine the degree of intersection with other selected cones or model faces. A cone can also be assigned various properties for the FE mesh.

Solid The model object *Solid* describes a solid body for the subsequent mesh generation with tetrahedron elements. Properties for the FE mesh can be assigned to the solid.

Model objects have no further significance once the element mesh is generated. However, they should be saved as drawings for reuse at a later time.

For an area to be meshed, a closed polygon must be defined using edges and/or walls. If you want to mesh multiple areas in different planes at the same time, then model faces should be defined as well.

Selecting

If you want to edit or delete an object, you first need to select it. This can be done in a number of different ways:

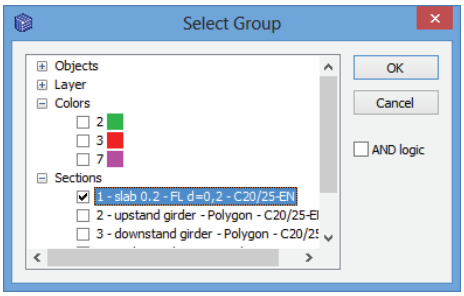
1. Select individual objects via mouse click. You can expand or reduce the selection by holding down the Shift or Ctrl key while clicking the objects.

2. Select an area by holding down the mouse button and dragging the cursor. When you let go of the mouse button, all objects completely inside the drawn rectangle will be selected.

Selecting with the right mouse button, the shortcut menu with the standard commands for the selected objects will appear.

3. The *Select* command in the *Edit* menu allows consecutive additions/deletions of the selection using selection boxes for example.

4. The *Select group...* command in the *Edit* menu allows you to select objects based on shared properties such as color, layer or section.



5. The *Select All* command in the *Edit* menu.

All of these selection methods can be combined with one another. Selections are removed when you click on an empty space in the workspace.

You can access the shortcut menu for objects that are already selected by holding down the Shift or Ctrl key and pressing the right mouse button.

Entering Coordinates

You can enter coordinates with either the keyboard or the crosshair.
If you are using the keyboard, enter the x, y and then z coordinates separated by a space in the dialog bar. The measures are always meter [m]. The positive direction corresponds to the displayed coordinate system.

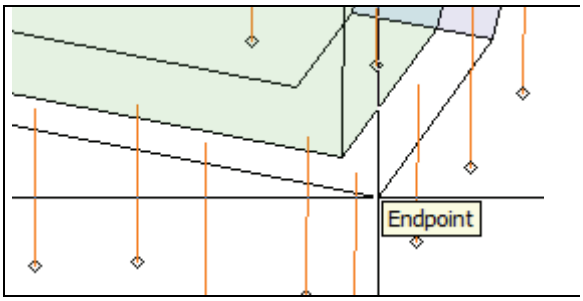
For 2D editing you only need to enter the first two coordinates. The program will automatically set the third coordinate to zero. The decimal separator (e.g., on the numeric keypad) conforms with the one defined in the regional settings.

Global coordinates These coordinates always refer to the global zero point.

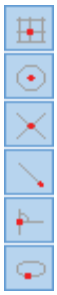
© InfoGraph GmbH, July 2015

Relative coordinates These coordinates refer to the last point that was entered (local zero point). A small axis system is always visible at this point. To activate relative coordinate input, click the *Relative* button.

Using the crosshair to enter coordinates, it automatically refers to an existing object.



The current snap option will be displayed.

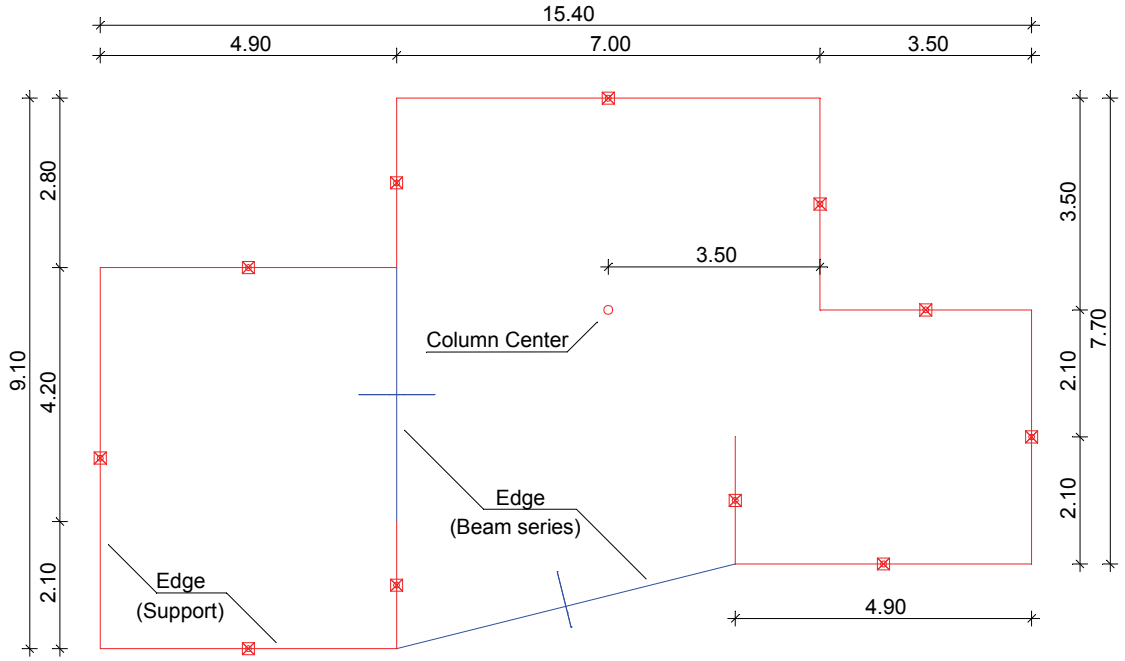


The *Node*, *Midpoint*, *Intersection*, *Endpoint*, *Perpendicular* and *Object* snap functions can be used to define the snap mode and ignore the automatic snap function. Click the appropriate button to activate them.

The snap window will then appear in the crosshair:
The selected snap function will remain active until it is disabled or replaced by another.

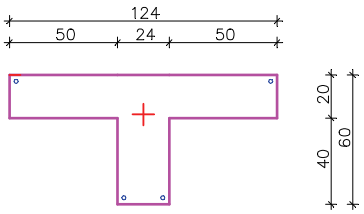
Example 1: Slab with Downstand Beam

This example shows you how to specify and calculate a basic floor plate and then process the results.



Material:	Concrete	C20/25-EN (Concrete EN 1992-1-1)
	Reinforcing steel	BSt 500, axis distance from edge 3.0 cm
Sections:	Slab	Thickness 20 cm

Downstand beam



Loads:	Permanent load	Dead load and additional load 1.50 kN/m²
	Traffic	Area load 3 kN/m²

A new project is started when you launch the program or select *New* from the *File* menu. Choose the ***Finite Elements*** structure type from the structure menu.

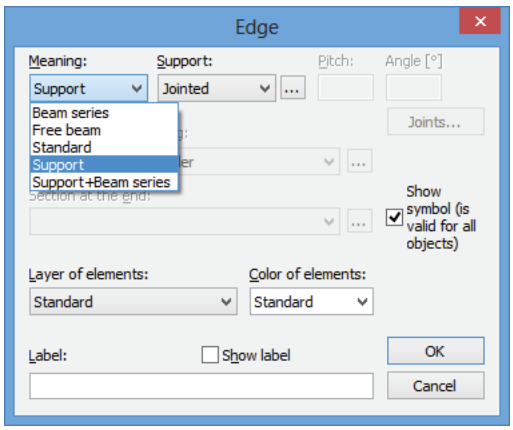
Drawing Ground Plans with Model Objects

Ground plans and in spatial structures wire frame models are initially described using the *Edge*, *Column* and *Hole* model objects. These objects will later be recognized by the mesh generator and taken into account during mesh generation.

You do not need to make any definitions regarding the area:

Simply draw your static system. The mesh generator will take care of the rest.

Click the *Edge* button and select properties in the dialog bar.



For *Meaning* set the option *Support* and leave the dialog with *OK*.

A crosshair will appear in the representation area and you will be prompted to enter the coordinates of where the edge starts in the dialog bar. Enter [0 0] to position the start of the edge at the origin.

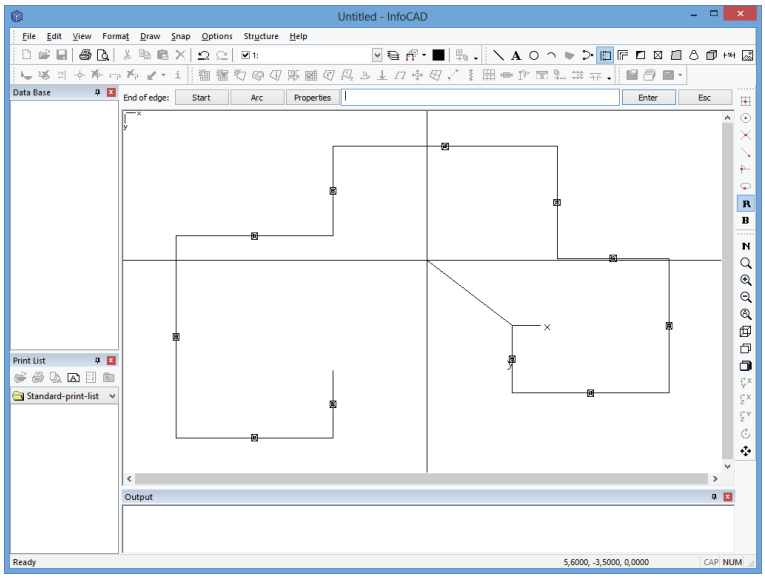
After you press [↵] or *Enter*, the end of the edge will be queried.

To enter additional coordinates, you should activate relative coordinate input by clicking the *Relative* button. A small axis system will now always appear at the last point inserted.
All successive coordinates you enter always refer to this point.
Enter [0 2.1] to define the end of the edge in the positive y direction. If you enter consecutive coordinates, the subsequent edge ends will always be queried. The remaining coordinates are therefore:

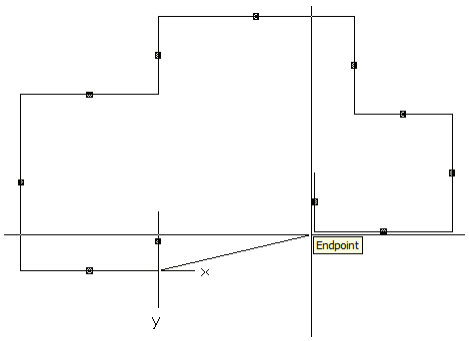
[-4.9 0] [0 –6.3] [4.9 0] [0 –2.8] [7 0] [0 3.5] [3.5 0] [0 4.2] [-4.9 0] [0 –2.1]

To temporarily enlarge the view, click *Zoom-* or *Zoom all*.

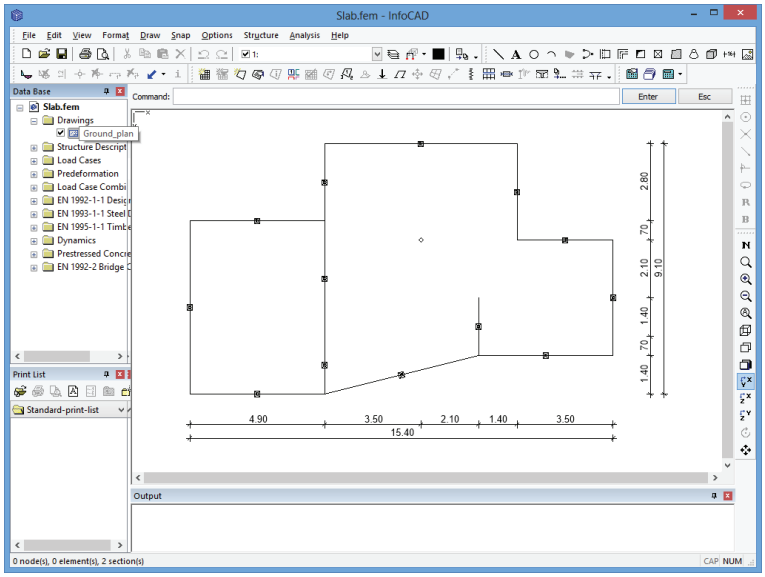
© InfoGraph GmbH, July 2015



You can begin entering another edge without having to use the command over again by selecting the *Start* option. Now enter the missing edges. In the process you can click on the endpoints of existing edges.



Click the *Column* button and enter the column midpoint. To move the current reference point for the entry of coordinates, click the *Basepoint* button and select the end of the edges to the right of the column as the new reference point. Confirm the *Old* or *Global* setting (the local reference system is not to be rotated) as the *New direction of x axis*.
The column should be 3.5 m in the negative x direction from the selected edge end point, which means the coordinates for the column midpoint are: [-3.5 0].



One way to check your input is to dimension the system. To do so, click the *Dimension* button and the *Objects* option. Now select all of the specified model objects. Dimension points are the end points of all objects. For example, to position a horizontal dimension chain, select *Horizontal* and click the drawing area below your system.

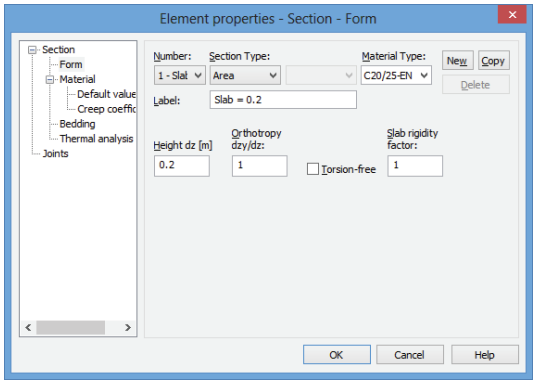
Click *Save* to store all the data you have entered up to this point. The Windows file dialog will appear and prompt you to enter a file name. The *.fem extension will be assigned automatically. Your drawing will also be given a name and is saved in this file.

Specifying Supports and Cross-Sections

The properties of the specified *Edge* and *Column* model objects are preset as follows:

Edge:	Support jointed
Column:	Support jointed

The correct support conditions are thus already set for this system.

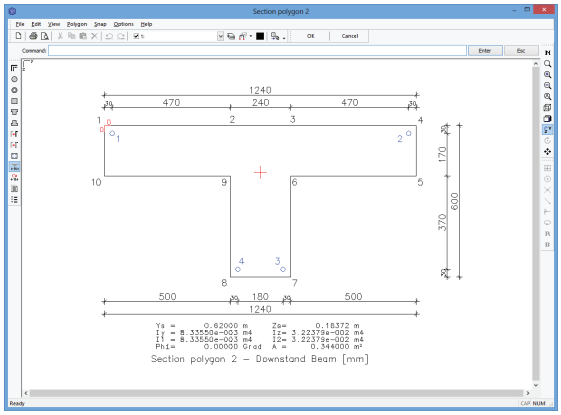


With the selection of the material the possible design standards are determined. The corresponding properties are now available.

For example the material C ..... -EN-D is to choose when the checks per NA Germany are to be performed.

For the other European standards, the material selection is C..... -EN.

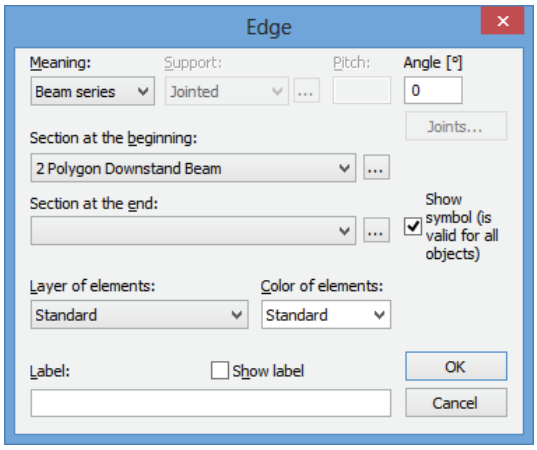
To modify the properties of the area elements, select the *Element Properties* command. Click *New* and then change the section type to *Area* for **Section 1**. The thickness is preset correctly with 0.2m and the material must be selected to C20/25-EN. For the next input this material is preset. The default settings for the remaining section properties such as bedding, reinforcing steel and EN 1992-1-1 are correct for this example.
Click *New* again and generate **Section 2**. Change the section type to *Polygon*. The input window for graphically describing the polygon is displayed:



You can generate and dimension the polygon shown above using the *Downstand Beam* command. Close the window by clicking *OK*.
Activate the *Reduce dead load* button to decrease the dead load of the T-beam by the portion attributed to the slab.

© InfoGraph GmbH, July 2015

Beam series (downstand beams) with the specified section should be generated at the two last entered edges to replace the supports. Open the *Edge* dialog by double-clicking the edge you want to modify. Now change the *Meaning* to *Beam series* and select Section 2. You do not need to specify the section at the end of the edge.



Generate the FEM Mesh

The finite element mesh can be generated automatically using the specified model objects. To do so, click the *Mesh Generation* button and select the *Form-sensitive* option. Arrow indicators will now appear marking the outer area limits recognized by the program. Specify the mesh width (0.45 in this example) to generate the FEM mesh, supports and beam series.

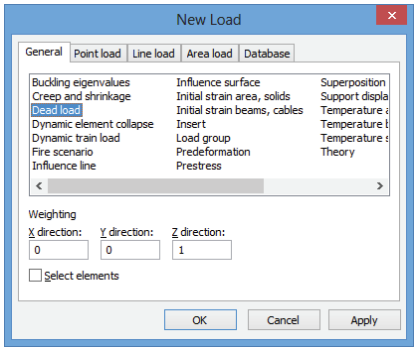
The drawing is not required anymore. Deactivate the drawing in the database

Defining Load Cases

Open the *Load cases* folder in the database and double-click

. A new load case will be created automatically and the load input and editing buttons will appear in the dialog bar.

Click the *Load creation* button to open the *New Load* dialog:

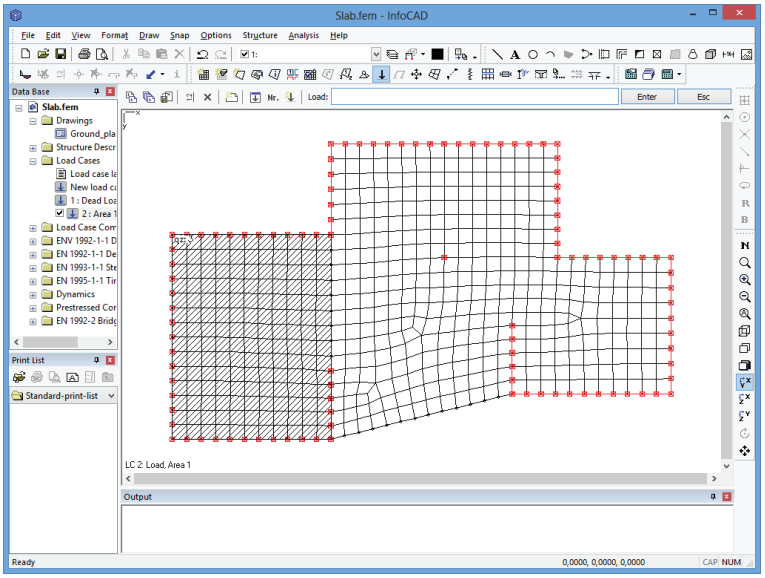


Select *Dead load* with a weighting of 1 in the Z direction and then click *Apply*. Using this load type, the dead load will be generated automatically based on the sections and material. This load is indicated by the *Dead Load* label in the upper right corner of the image.

Now select *Uniform region* on the *Area load* tab and select two points to specify a rectangle that surrounds the entire slab. The load ordinate qz should correspond to 1.5 kN/m². This load will be applied to all elements located entirely within the area.

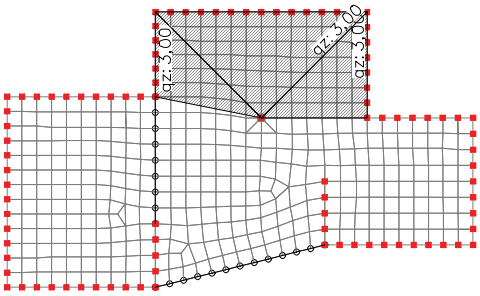
Click *Change Number* to modify the load case number or to specify a load case name.

Create the new load case 2 using the function *New load case*. Select the *Uniform rectangle* area load under the *Load creation* option. The load rectangle is now represented by three coordinates. Click on the three corner points of the load field. In this case the load ordinate is qz = 3 kN/m²:

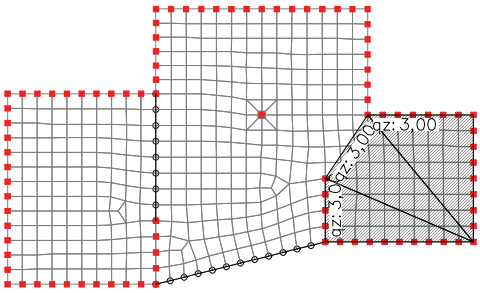


Create the next load case and use the *Uniform polygon* area load for each remaining field. Trace the next field and then click *Close* and enter the load ordinate qz = 3 kN/m². The program will then apply triangle areas to the defined load area.

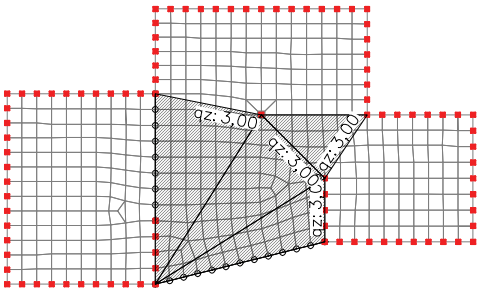
Now enter the depicted load cases in consecutive order, each time selecting *New load case:*



Load case 3



Load case 4



Load case 5

Load input is now complete. Exit the load dialog by clicking *Close* (in the dialog bar).

© InfoGraph GmbH, July 2015

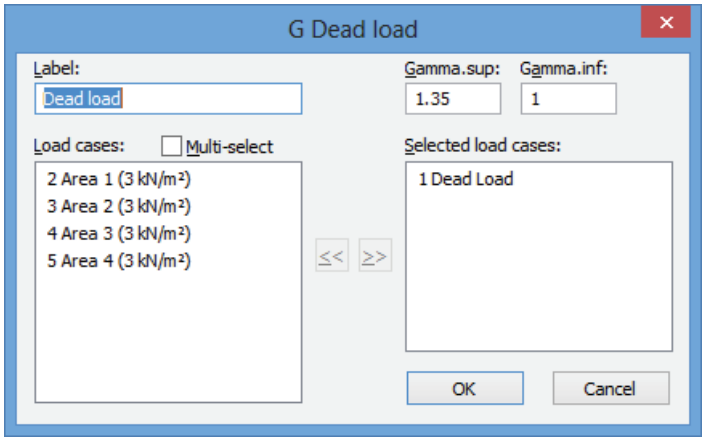
Specifying Actions and Design Situations

When carrying out the design, first the internal forces of the actions are combined. The program uses cyclic permutation to analyze all variations of the leading and accompanying actions. The safety factors and combination coefficients are automatically taken into account as specified by EN 1990. The extremal internal forces are then used to determine the design values.

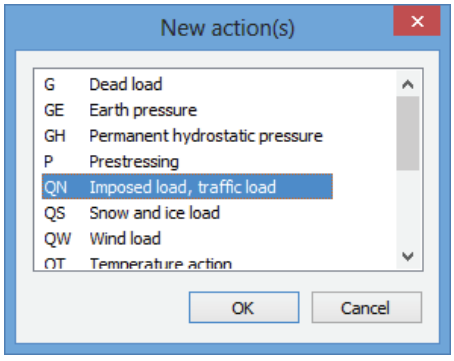
This is done by simply assigning the existing load cases to the actions:

G	Dead load	Load case 1
QN	Imposed load, traffic load	Load case 2–5

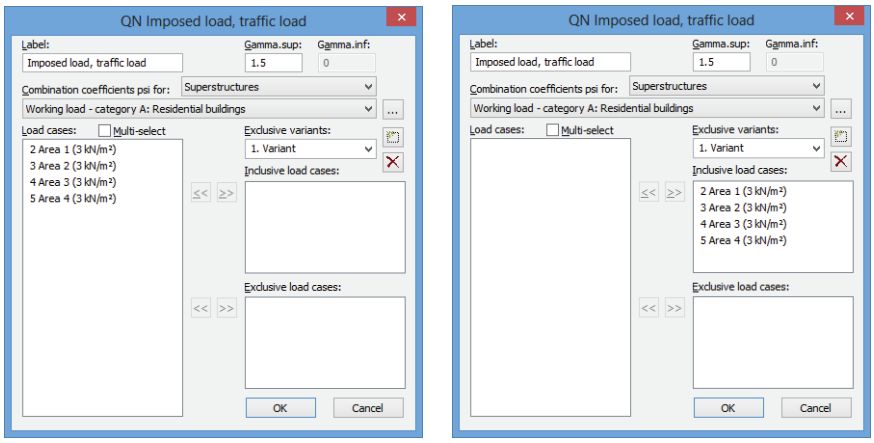
Open the *EN 1992-1-1 design* folder in the database and select the *Actions* submenu item. A dialog will appear where you can define actions in accordance with EN 1992-1-1. The program suggests a new *G Dead load* action. Confirm by clicking *OK*.
Now assign the relevant load cases to the action by selecting a load case in the left pane and moving it to the *Selected load cases* pane on the right:



Exit the *Dead load* dialog and create a new *QN Imposed load, traffic load* action.



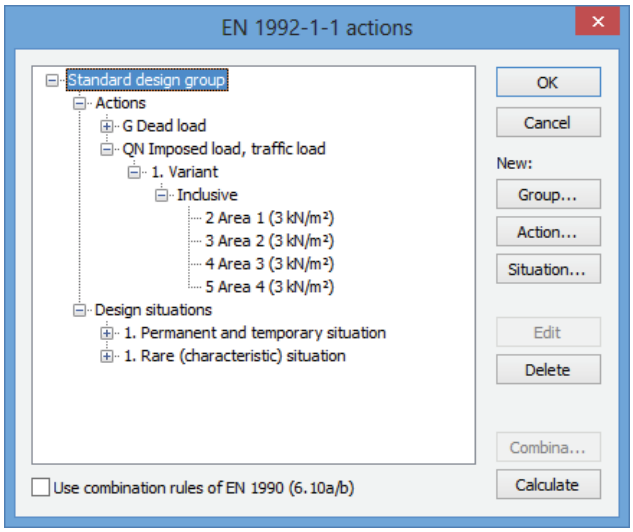
Assign the remaining traffic loads by moving them to the *Inclusive load cases* area.



Inclusive load cases	Selected load cases that can have a simultaneous effect.
Exclusive load cases	Selected load cases that are mutually exclusive (e.g., wind from right / left or individual mobile load positions)

The action QN is now defined and you can exit the dialog.

Next click *Situation...*, select *Permanent and temporary situation* and then accept the default settings. The *Rare (characteristic) situation* is required to define the robustness reinforcement. Create this as well.



The actions and situations are now defined. You can edit an action or situation by double-clicking it. The user-friendly tree structure provides an easy way of checking the included load cases.

Click *OK* to exit the *EN 1992-1-1 actions* dialog.

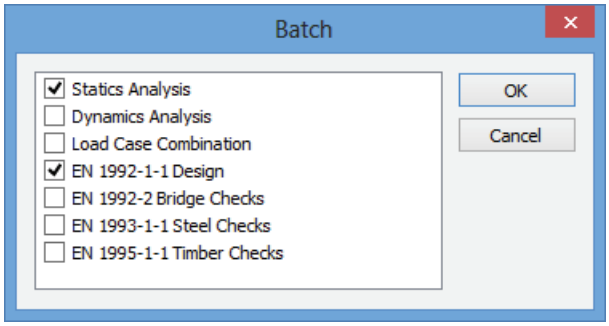
© InfoGraph GmbH, July 2015

Performing Calculations

You can now perform the following calculations:

- *Statics analysis*
  (determines deformations, internal forces, support forces, etc., for each load case)
  - *EN 1992-1-1 design* (bending and shear design)

The calculations can be started automatically in consecutive order. To do so, click the *Batch...* button and select the relevant check boxes.



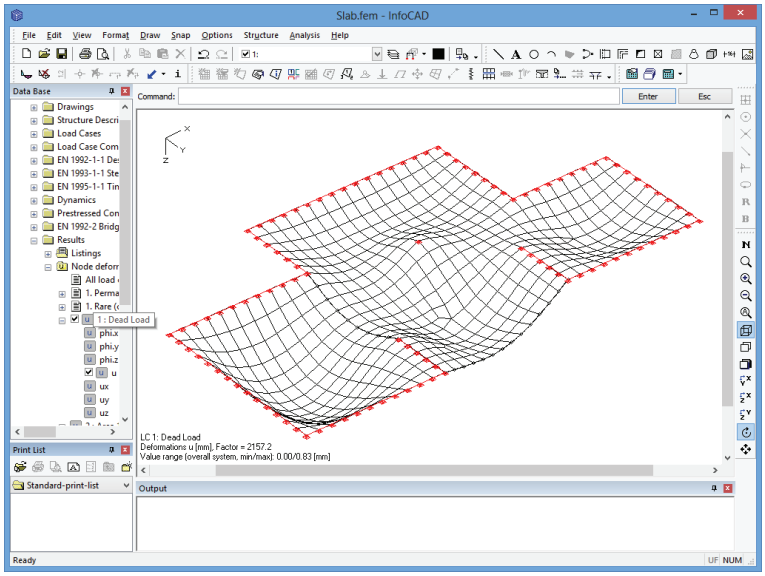
If necessary, you can control the calculation and logging process by clicking the *Calculation settings...* menu item.

The progress of the calculation and any errors or warnings will be displayed on the output bar.

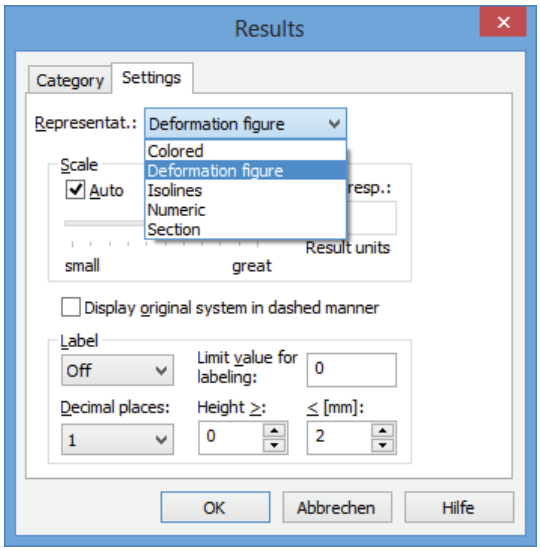
Processing Results

All of the available results are listed in the *Results* folder in the database. Double-click a result to open the available or designated standard view (graphic or table) of that result.

For instance, open the *Node deformations* results folder and then double-click *Load case 1*. The deformed system will be displayed. To see the deformation figure, switch to the *3D* view.



Click the *Results* button in the results bar to open a dialog that lets you select the display option and settings for all results:



If you select representation *Section*, you can click the *Define section* button in the results bar and define the sections for the results view.

Please note that some view options are not available depending on the category you selected:

- A deformation figure can only be displayed for all components simultaneously and not for an individual displacement component (e.g., uz).
- The min/max line from an internal forces combination of area elements can only be displayed numerically or in the section.

EN 1992-1-1 Checks

In addition to the bending and shear design that has already been carried out, you can also perform additional checks in the ultimate limit state and the serviceability limit state:

- Minimum reinforcement for securing ductile member behavior (robustness)
- Pure torsion and torsion with lateral force
- Checks against punching shear
- Check for concrete and reinforcing steel fatigue (bending, shear and torsion)

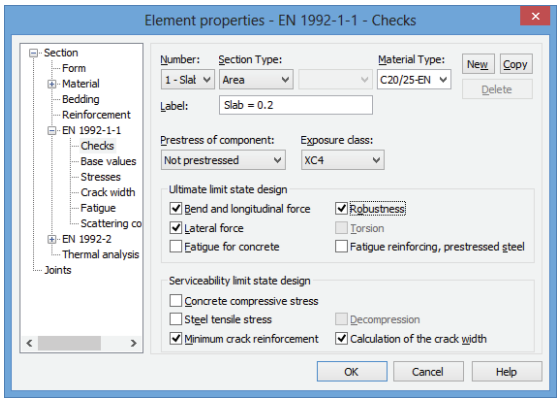
- Limiting the concrete compressive stresses
- Limiting the reinforcing steel stresses
- Minimum reinforcement for the crack width limitation
- Limiting the crack width via direct calculation

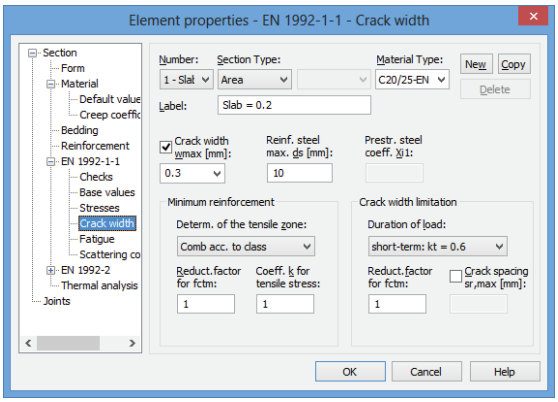
© InfoGraph GmbH, July 2015

Additional input is required in the *EN 1992-1-1 design* folder for this purpose:

Actions: Define the necessary design situation depending on the additional checks and the exposure class.

Settings related to the section are made in the *Element properties* dialog. Click the *Element properties* button to open this dialog and then enter the check specifications required for each section under *EN 1992-1-1*.





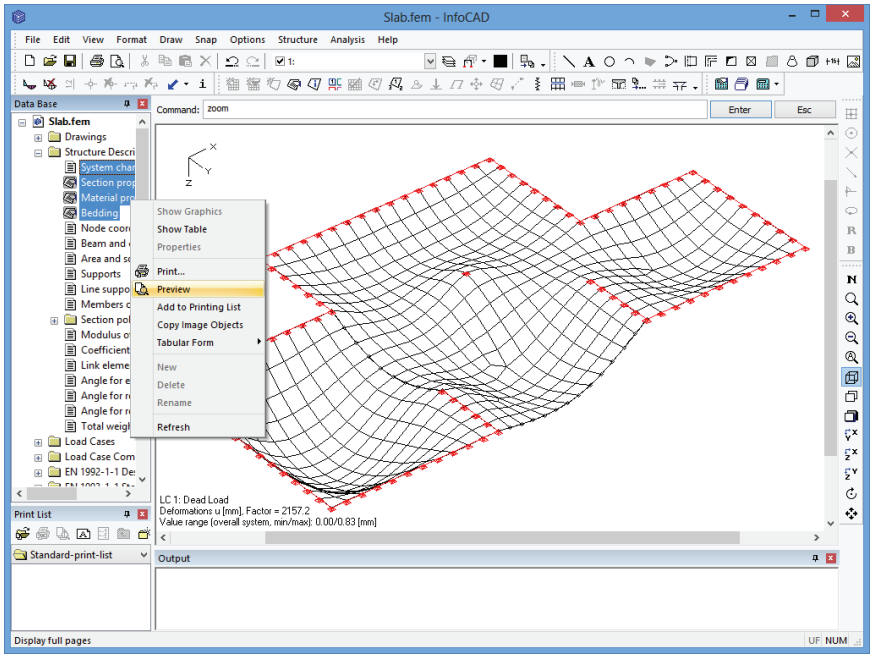
After you are done, start the *EN 1992-1-1 design*. The program will determine the required reinforcements, perform the checks and then generate a log.

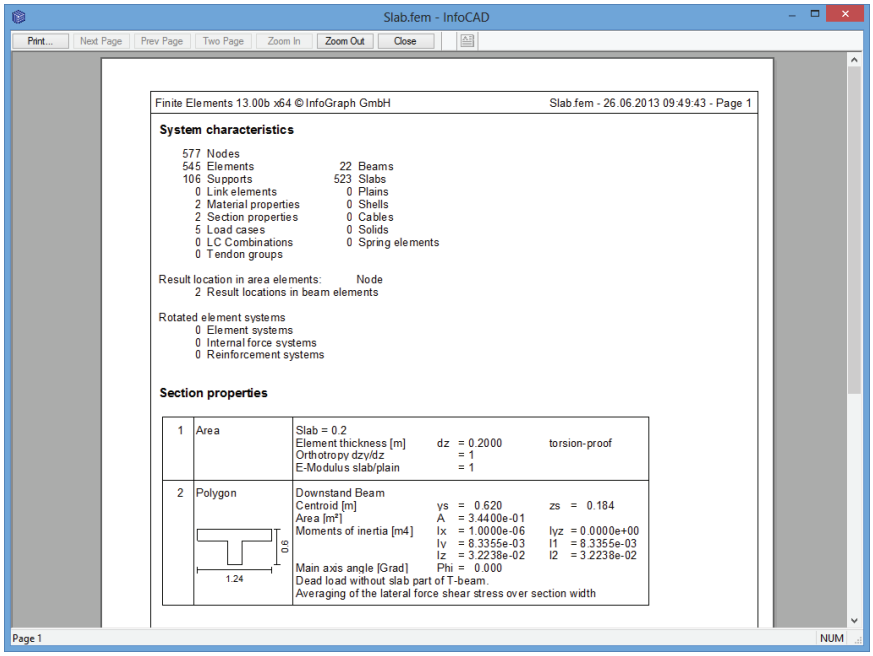
A detailed description of all the checks is available in the InfoCAD help and in the user manual.

Printing

You always have the option of printing the current screen view. Click *Print Preview* to see what the actual printout will look like. The settings for the page format, printer selection, etc., can be modified using the *Page Setup...* command in the *File* menu.

Tables or logs need to be selected in the database and then printed out from the shortcut menu (right-click). The *Page Preview* command is also available in the shortcut menu.





© InfoGraph GmbH, July 2015

Printing List

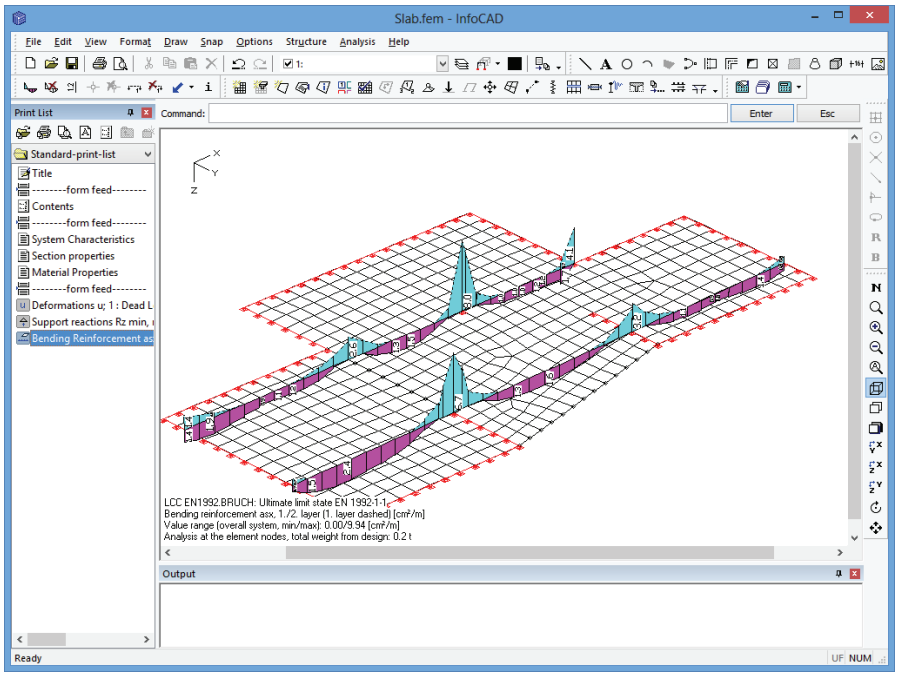
Specified views as well as tables and logs can be added to the printing list for printing at a later time. Only display parameters are stored, meaning changes made to the system are automatically reflected in the list.

Switch from the database to the printing list. The *Print* and *Page Setup* commands can also be accessed in the printing list.

Click the *Add to printing list* button to copy the current system view into the printing list. All active settings will be saved with the system view.

Use the shortcut menu to add tables or logs selected in the database to the printing list. You can also add separate text objects, page breaks or a table of contents.

You can sort existing entries in any order you want by simply dragging them with the mouse.



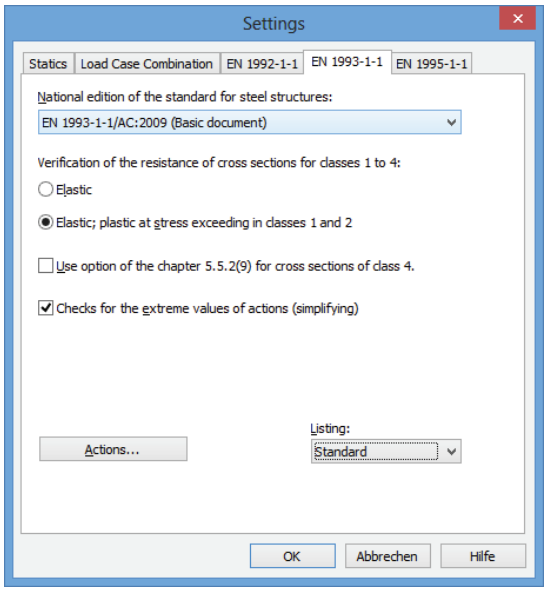
Double-click entries in the printing list to open the corresponding view in the representation area.

Example 2: 2D Hall Frame

This example shows you how to specify a 2D hall frame and perform the following calculations:

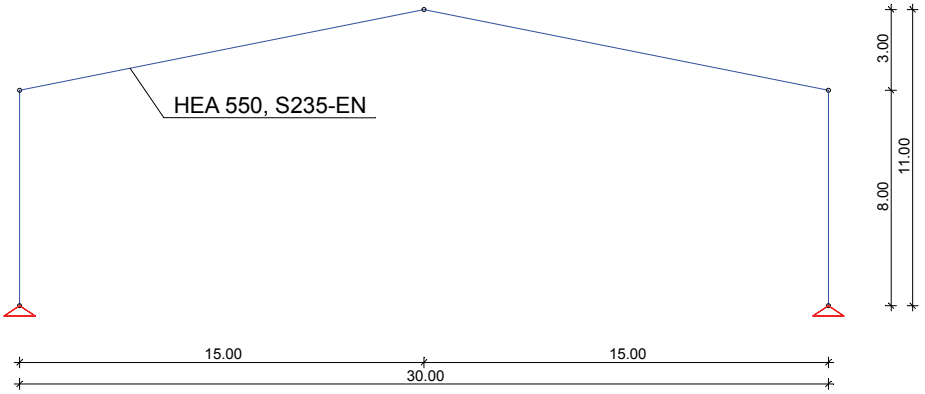
- Static calculation
- EN 1993-1-1 Steel Checks (elastic; plastic at stress exceeding in classes 1 and 2)

Please select in the menu *Analysis / Settings ...* EN 1993-1-1 the required national annex of the standard. This will be used in the subsequent inputs and calculations.
Activate the *Elastic; plastic at stress exceeding in classes 1 and 2* - button.



In accordance with Chapter 6.2.1 of the standard, the elastic cross-section resistance is verified for classes 1 to 4.
If the comparison stress in classes 1 and 2 exceeds the permissible limit, the plastic cross-section resistance will be verified. For every set of internal forces the cross-section class is automatically determined.

Task



A new project will be started when you launch the program or select *New* from the *File* menu. Choose the ***2D Frame*** structure type you want to use from the structure menu.
The representation area will switch to the x-z view and you can now start specifying the system.

© InfoGraph GmbH, July 2015

Specifying the Framework

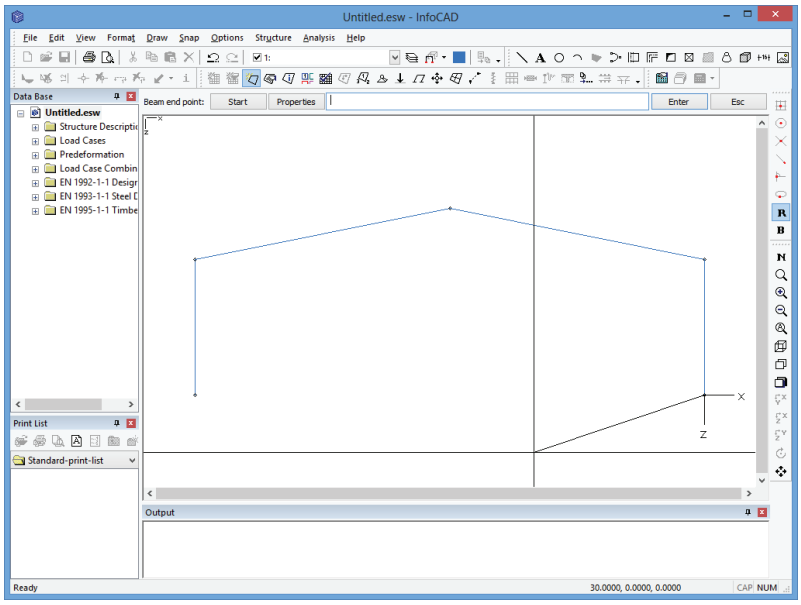
Click the *Element creation* button and a crosshair will appear in the representation area. You will then be prompted to enter the coordinates of the beam starting point in the dialog bar. Enter [0 0] to position the first beam node at the global origin. Press [↵] or *Enter*. Next, the beam end point will be queried.

To enter additional coordinates, you should activate relative coordinate input by clicking the *Relative* button. A small axis system will now always appear at the last point inserted. All successive coordinates you enter always refer to this point. Enter [0 -8] to define the beam end in the negative z direction. You have now specified the first beam.

To temporarily enlarge the view, click *Zoom* or *Zoom all*.

Enter the remaining beam ends in consecutive order and adjust the view where necessary:

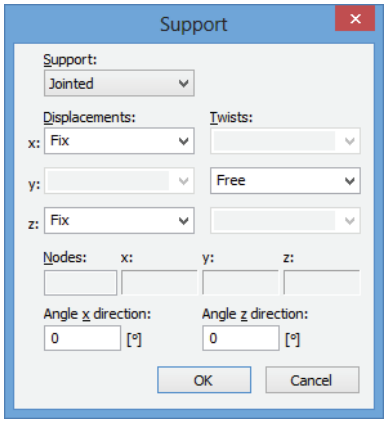
[15 -3] [15 3] [0 8]

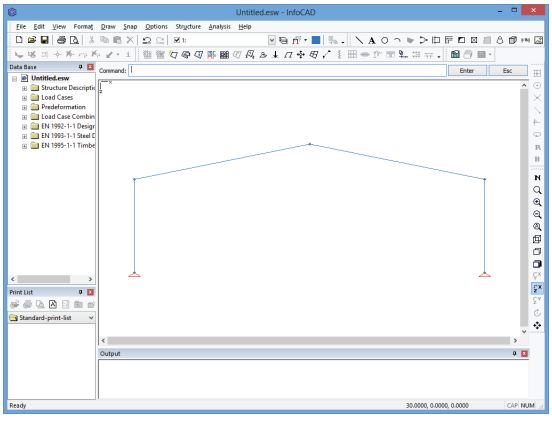


Exit beam input by pressing [↵] or *Enter*.

Now click the *Supports* button and choose the *Create* option. You will be prompted to select the beam nodes in the dialog bar. You can click the nodes directly with the crosshair or select them using the *Window* option (to specify a rectangular snap window). Press *Enter* after you have selected the nodes. Specify *Yes* when the program asks you for the global alignment.

In the *Support* dialog you can individually adjust each degree of freedom if necessary.



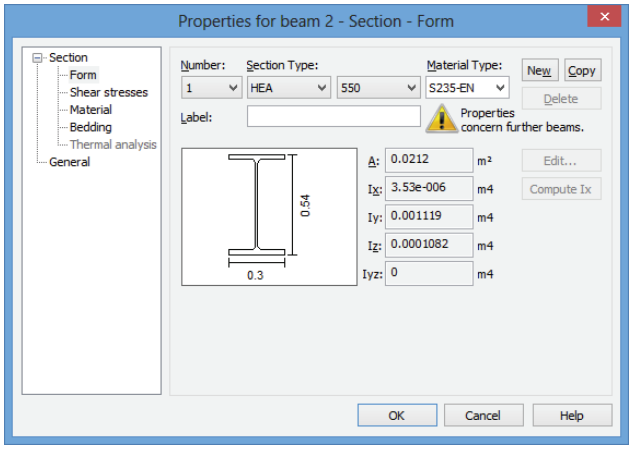


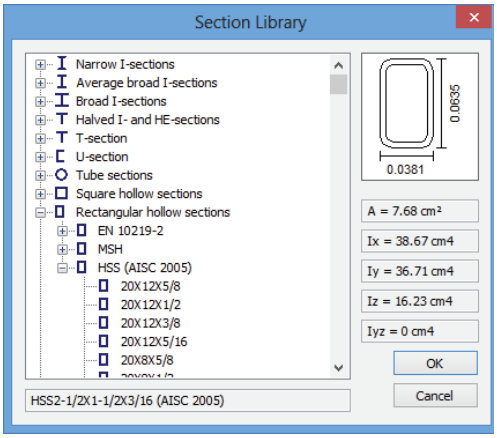
Confirm that the support is set to *jointed* and exit the *Support* dialog by pressing *OK*. The supports are indicated by their corresponding icons.

Click *Save* to store all the data you have entered up to this point. The Windows file dialog will appear and prompt you to enter a file name. The *.esw file extension will be added automatically.

Defining the Section

Click the *Element properties* button and set the section type to *HEA 550* and the material type to S235-EN for section 1.





You also have access to a detailed section library with international steel profiles.

The complete framework is now defined and the corresponding parts list is available in the workspace in the *Structure description* folder.

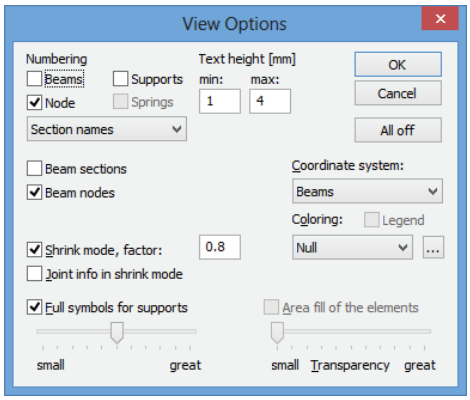
List of steel profiles

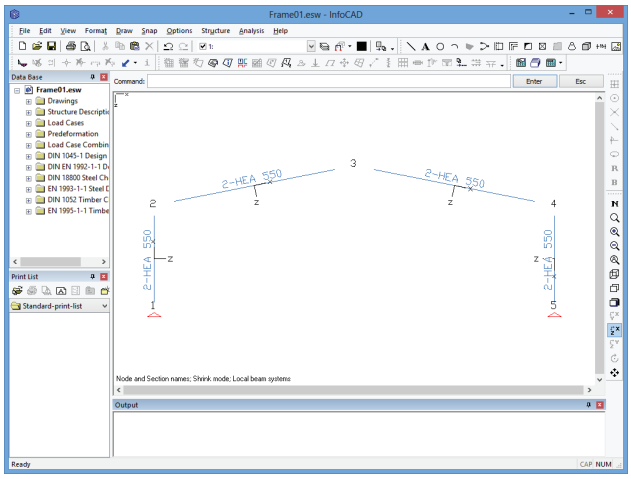
	Section	Length [m]	Quanti ty	Total [m]	Weight [kg/m]	Weight [kg]	Total [t]
1	2 HEA 550 - S235-EN	8.000	2	16.000	166.420	1331.36	2.66
2	2 HEA 550 - S235-EN	15.297	2	30.594	166.420	2545.74	5.09
3	Sum		4	46.594			7.75

Checking the Structure Properties

You can activate a number of different view options to check the data you have entered. To do so, click the *View options* button:

© InfoGraph GmbH, July 2015

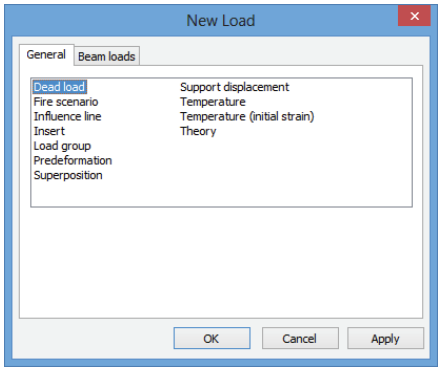




Defining Load Cases

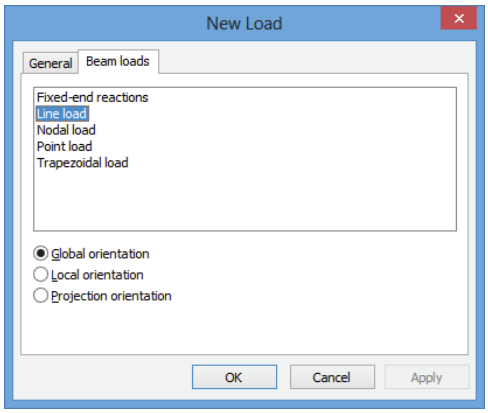
Open the *Load cases* folder in the database and double-click New load case. A new load case will be created automatically and the load input and editing buttons will appear in the dialog bar.

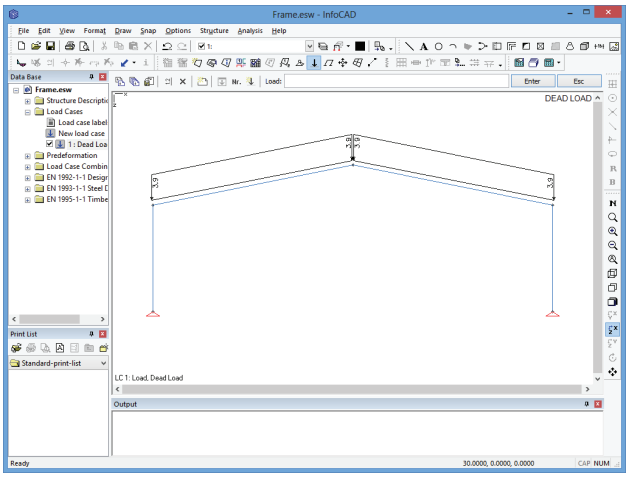
Click the *Load creation* button to open the *New Load* dialog:



Activate the *Dead load* and click *Apply*. Using this load type, the dead load will be generated automatically based on the sections and material. This load is indicated by the *Dead Load* label in the upper right corner of the image.

Now choose *Line load* from the *Beam loads* tab, check the *Global orientation* option, select both beams of the horizontal frame member and enter a load ordinate of 3.9 kN/m:

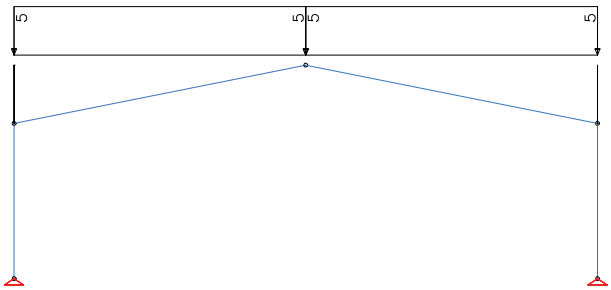




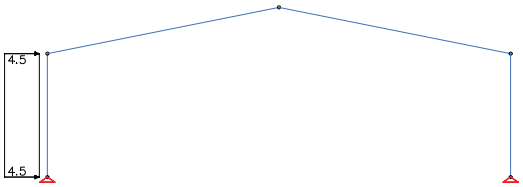
You can adjust the load diagrams in any manner you want using the *Load representation* button.

Click *Change Number* to modify the load case number or to specify a load case name.

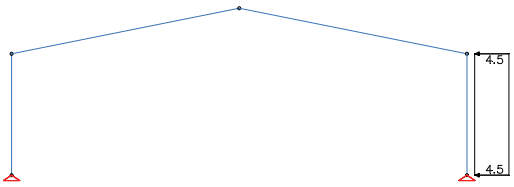
Create the new load case 2 by pressing New load case. Choose *Line load* from the *Load creation* tab, check the *Projection orientation* option and enter a snow load of 5 kN/m.



Starting with *New load case*, enter the following load cases:

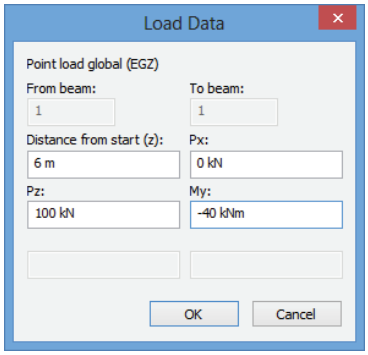


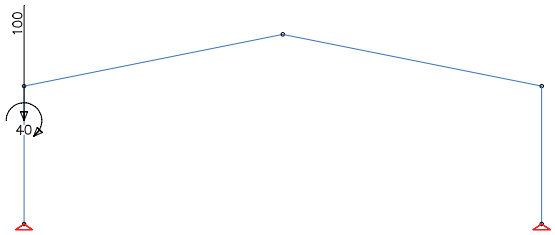
Load case 3: wind from the left



Load case 4: wind from the right

The 'Load case 5: crane load left' requires a single load to be positioned inside the frame column. This load should be applied 2 meters below the frame corner.
After selecting the *Point load* load type, simply click the corresponding frame column and enter the ordinates as shown below:





For precise positioning, define a load location with a distance of 6 meters to the beam start.

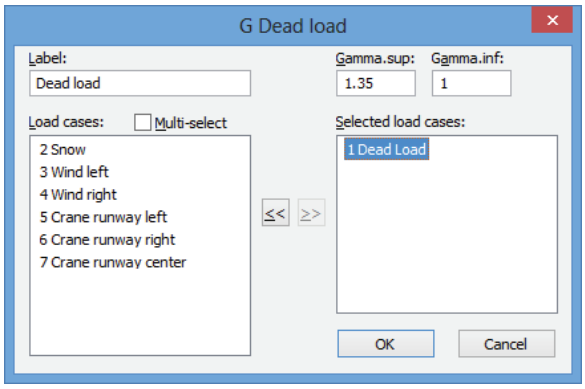
Now enter 'Load case 6: crane load right' (distance of 2 meters to beam start) and 'Load case 7: crane load middle' accordingly, with half the load on each frame column.

EN 1993-1-1 Steel Checks

You first need to describe the actions according to EN 1993-1-1. To do so, assign different actions to the existing load cases:

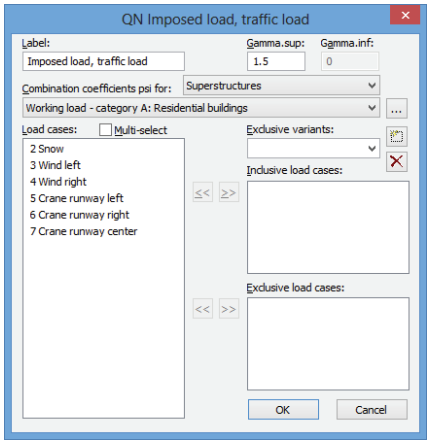
© InfoGraph GmbH, July 2015

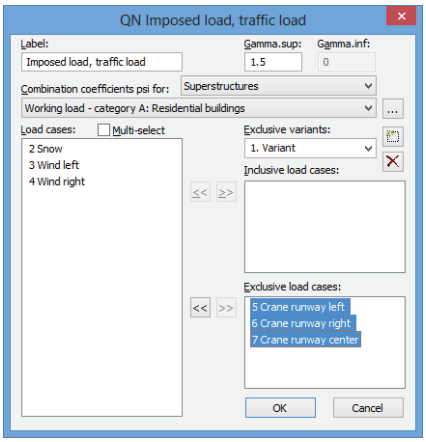
G	Dead load:	Load case 1
QS	Swow ans ice load:	Load case 2
QW	Wind load:	Load case 3 and 4
QN	Imposed load, traffic load:	Load case 5, 6 and 7



Open the *EN 1993-1-1 Steel Checks* folder in the database and select the *Actions* submenu item. A dialog will appear where you can define actions in accordance with EN 1993-1-1. The program suggests a new *G Dead load* action. Confirm by clicking *OK*.
Now assign the relevant load cases to the action by selecting a load case in the left pane and moving it to the *Selected load cases* pane on the right.

Exit the *Dead load* dialog and create a new *QN Imposed load, traffic load* action. Assign the remaining traffic loads by moving them to the *Exclusive load cases* area.

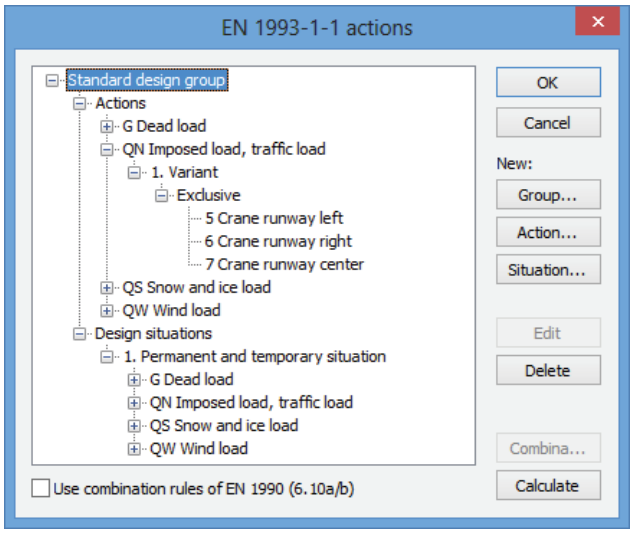




Inclusive load cases	Selected load cases that can have a simultaneous effect.
Exclusive load cases	Selected load cases that are mutually exclusive (e.g., wind from right / left or individual mobile load positions)

Do the same with the *QS Snow and ice load* and the *QW Wind load* action.

Next click *Situation...*, select *Fundamental combination* and then accept the default settings.



The actions and situations are now defined. You can edit an action or situation by double-clicking it. The user-friendly tree structure provides an easy way of checking the included load cases.

Click *OK* to exit the *EN 1993-1-1 actions* dialog.

Performing Calculations

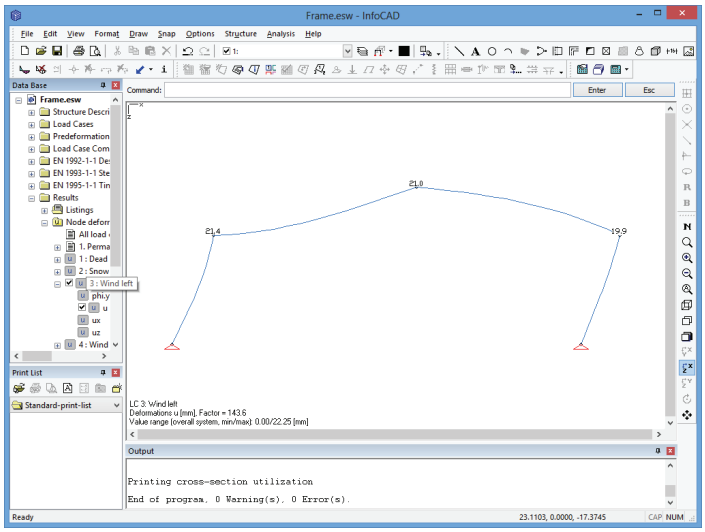
You can now start the following calculations:

- Static analysis (determines deformations, internal forces, support forces, etc., for each load case)
  - EN 1993-1-1 Steel Checks (superposition of actions with calculation of determinant design values, check for elastic or plastic utilizations)

The calculations can be started automatically in consecutive order. To do so, click the *Batch...* command in the calculation menu and activate the relevant check boxes.

The progress of the calculation and program messages, such as errors or warnings, will be displayed in the output bar.

Processing Results

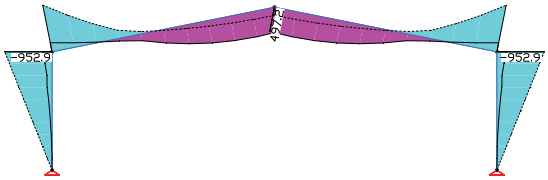


All of the available results are listed in the *Results* folder in the database. Double-click a result to open the available or designated standard view (graphic or table) of that result.
For instance, open the *Node deformations* folder and then double-click *Load case 3*. The deformed system will be displayed.

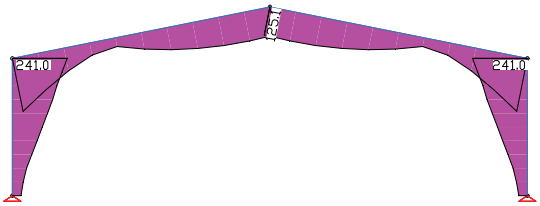
Click the *Results* button to access various display settings.

© InfoGraph GmbH, July 2015

Internal forces and comparison stresses from the fundamental combination:

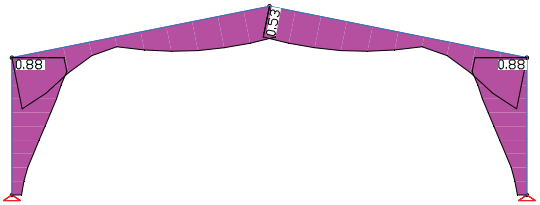


LCC EN1993.SV.1: 1. Permanent and temporary situation, EN 1993-1-1
Internal forces min,max My. 771.42 [kNm] =
Value range (overall system, min/max): -952.91/513.35 [kNm]

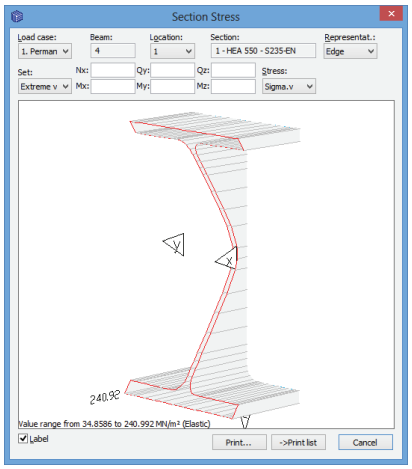


LCC EN1993.SV.1: 1. Permanent and temporary situation, EN 1993-1-1
Stresses (steel checks) max Sigma.v. 195.09 [MN/m²] =
Value range (overall system, min/max): 37.91/240.99 [MN/m²]

You can activate the calculated utilizations in the *Stresses / Steel Checks* folder. Note that the depicted frame can only be checked if the plastic internal force limits are taken into account.



Utilization per EN 1993-1-1. 0.75 [-] =
Value range (overall system, min/max): 0.16/0.93 [-]

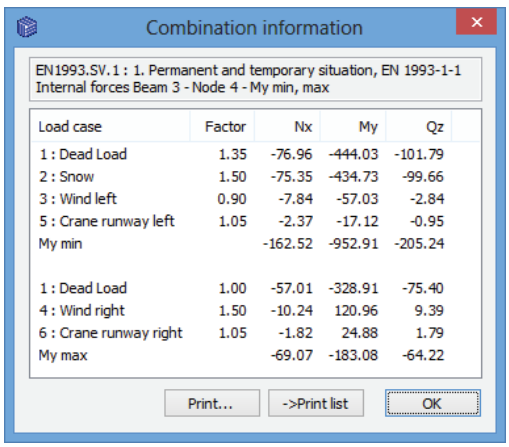


In the stress view right-click any beam and select *Section Stress* from the shortcut menu.

The stresses at the section will now be applied for the selected location.

In the depicted window you can select the results for all load cases and combinations as well as for the σx, σv, τx, τxy and τxz stresses.

The checking program records the calculation process in a concise log. You can view the log in the *Listings* folder.

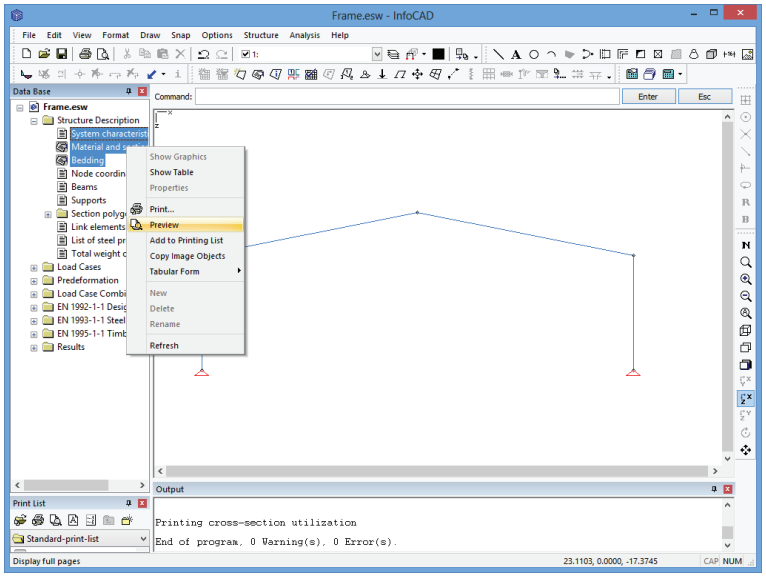

Combination information
EN1993.SV.1 : 1. Permanent and temporary situation, EN 1993-1-1
Internal forces Beam 3 - Node 4 - My min, max
Load case | Factor | Nx | My | Qz
1 : Dead Load | 1.35 | -76.96 | -444.03 | -101.79
2 : Snow | 1.50 | -75.35 | -434.73 | -99.66
3 : Wind left | 0.90 | -7.84 | -57.03 | -2.84
5 : Crane runway left | 1.05 | -2.37 | -17.12 | -0.95
My min | | -162.52 | -952.91 | -205.24
1 : Dead Load | 1.00 | -57.01 | -328.91 | -75.40
4 : Wind right | 1.50 | -10.24 | 120.96 | 9.39
6 : Crane runway right | 1.05 | -1.82 | 24.88 | 1.79
My max | | -69.07 | -183.08 | -64.22
Print...
->Print list
OK

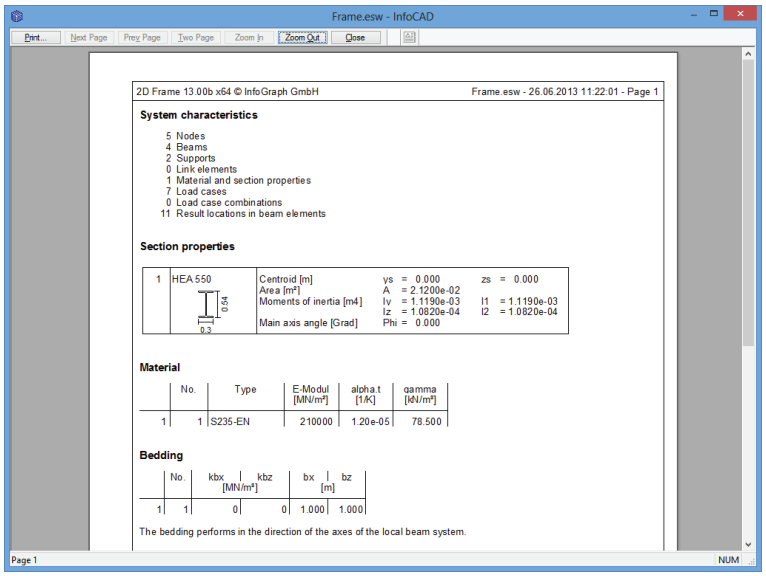
The combination information shows you the individual factors contributing to an internal force result.
In the results view right-click the desired results location in the beam and select *Combination information* from the shortcut menu.

Printing

You always have the option of printing the current screen view. Click *Print Preview* to see what the actual printout will look like. The settings for the page format, printer selection, etc., can be modified using the *Page Setup...* command in the *File* menu.

Tables or logs need to be selected in the workspace and then printed out from the shortcut menu (right-click). The *Page Preview* command is also available in the shortcut menu.




Frame.esw - InfoCAD
Print...
Next Page
Prev Page
Two Page
Zoom In
Zoom Out
Close
2D Frame 13.00b x64 © InfoGraph GmbH
Frame.esw - 26.06.2013 11:22:01 - Page 1
System characteristics
5 Nodes
4 Beams
2 Supports
0 Link elements
1 Material and section properties
7 Load cases
0 Load case combinations
11 Result locations in beam elements
Section properties
1 HEA 550
Centroid [m] ys = 0.000 zs = 0.000
Area [m²] A = 2.1200e-02
Moments of inertia [m4] Iy = 1.1190e-03 I1 = 1.1190e-03
Iz = 1.0820e-04 I2 = 1.0820e-04
Main axis angle [Grad] Phi = 0.000
Material
No. Type E-Modul [MN/m²] alpha.t [1/K] gamma [kN/m³]
1 1 S235-EN 210000 1.20e-05 78.500
Bedding
No. kbx kbz [MN/m²] bx bz [m]
1 1 0 0 1.000 1.000
The bedding performs in the direction of the axes of the local beam system.
Page 1
NUM

Printing List

Specified views as well as tables and logs can be added to the printing list for printing at a later time. Only references to the specified display parameters are stored, meaning changes made to the system are automatically reflected in the list.

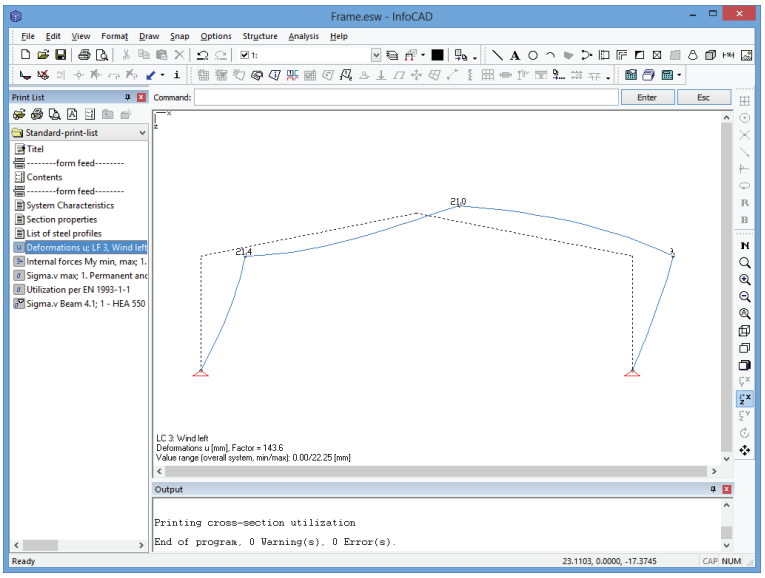
Switch from the database to the printing list. The printing list has its own *Print* and *Page preview* commands.

Click the *Add to printing list* button to copy the current system view into the printing list. All active settings will be saved with the system view.

© InfoGraph GmbH, July 2015

Use the shortcut menu to add tables or logs selected in the database to the printing list. You can also add separate text objects, page breaks or a table of contents.

You can sort existing entries in any order you want by simply dragging them with the mouse.

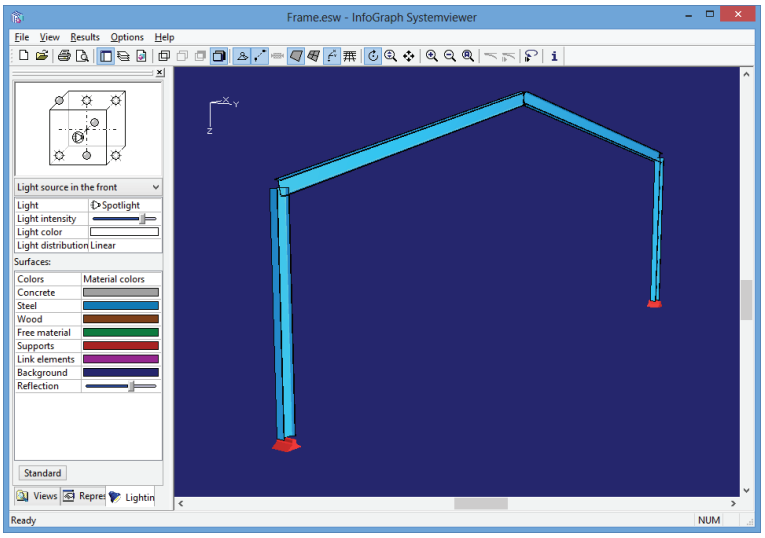


Double-click an entry in the printing list to open the corresponding view in the representation area.

InfoGraph Systemviewer

The InfoGraph Systemviewer allows you to visualize FEM and framework projects as well as to animate deformations from static or dynamic calculations.

Launch the Systemviewer from the *Programs / InfoGraph* folder in the *Start* menu and open the file you want to view. A variety of settings options are available to help you optimize the graphical output.



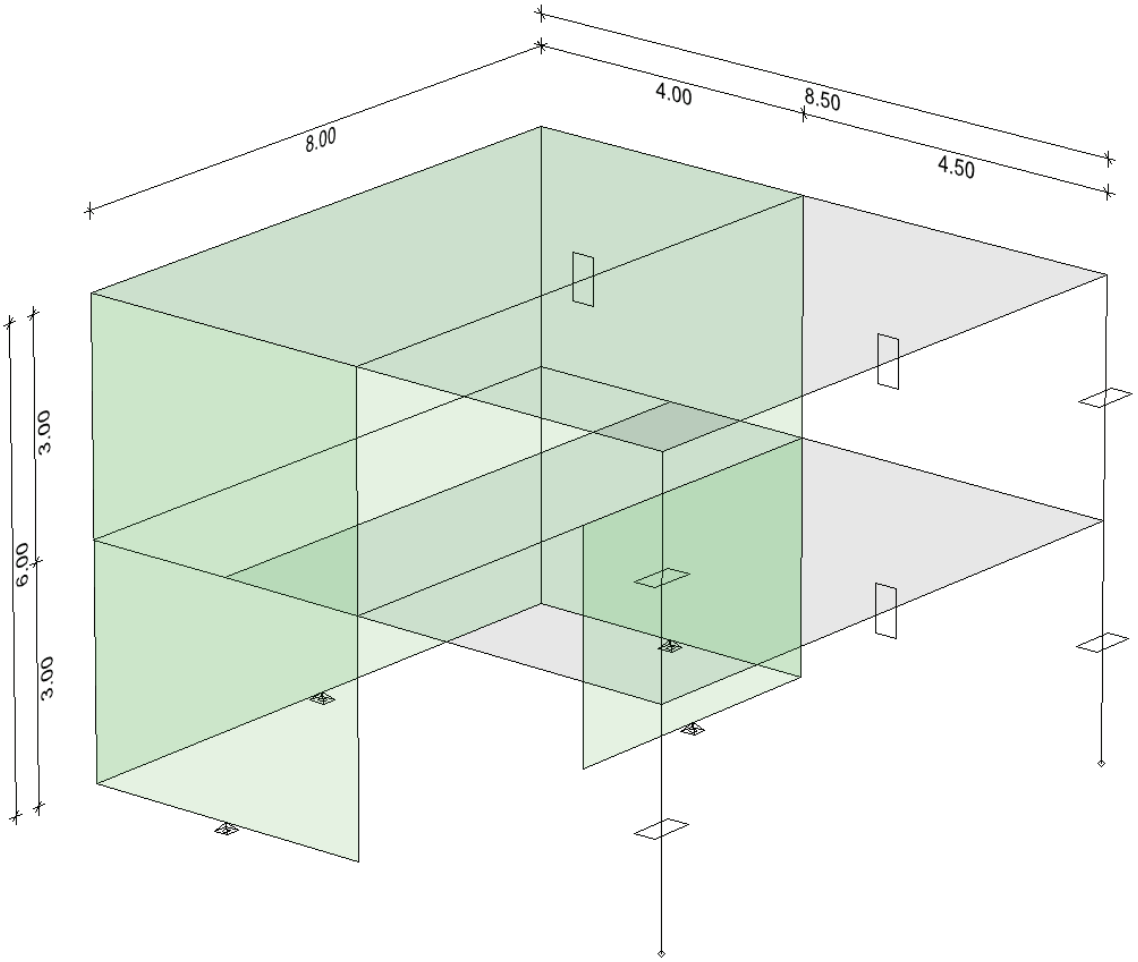
Example 3: 3D Building

This example shows you how to specify a 3D building and perform the following calculations:

- Static calculation
- Design according to EN 1992-1-1

It is assumed that you have already gone through the previous examples. Hence, some of the processing steps will not be detailed in this example.

Task



System axes with dimensions

Material	C20/25-EN (Concrete EN 1991-1-1) Reinforcing steel BSt 500, axis distance of 3.0 cm from edge
Load	Dead weight, area load, horizontal load

Procedure to enter 3D structures:

1. Describe a wire frame model using edges
2. Assign the wanted properties to the edges (support, beam series, free beam)
3. Enter model faces
4. Mesh generation

 © InfoGraph GmbH, July 2015

Drawing the wire frame model

Launch the program or start a new project by clicking *New* and then selecting ***Finite Elements*** under the *Structure* menu.

Click the *Edge* button and begin by entering:

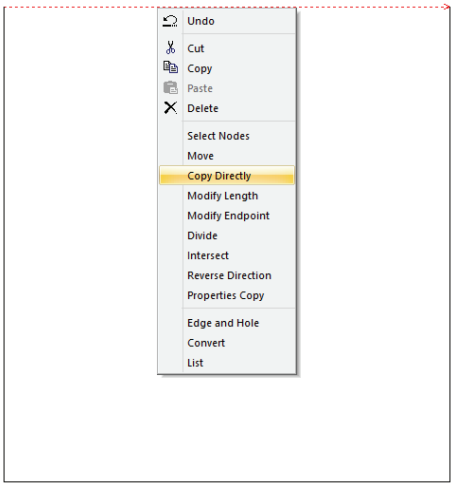
[0 0]

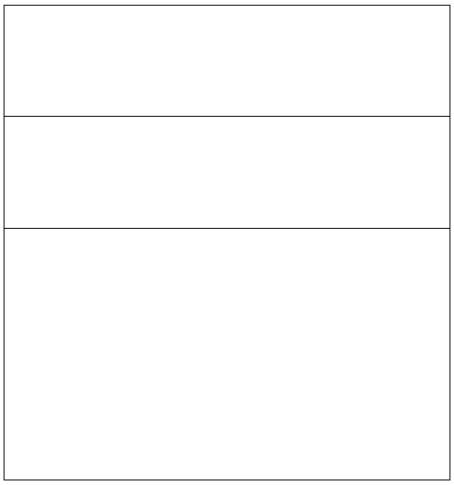
Activate relative coordinate input and complete the outer edge of the 1st level:

[8 0], [0 8.5], [-8 0]

Point to the start of the 1st edge to connect the lines.

Select the upper edge with the right mouse button and choose *Copy directly* from the shortcut menu.





The program asks for the displacement vector. Select e.g. the upper left end of the edge to define the start point. Now activate the option *Multiple*. For the second point of the displacement vector enter the coordinates [0 2] and [0 4]. The edge will be copied 2 and 4 meters in y direction. Finish the copy process with *Enter*.

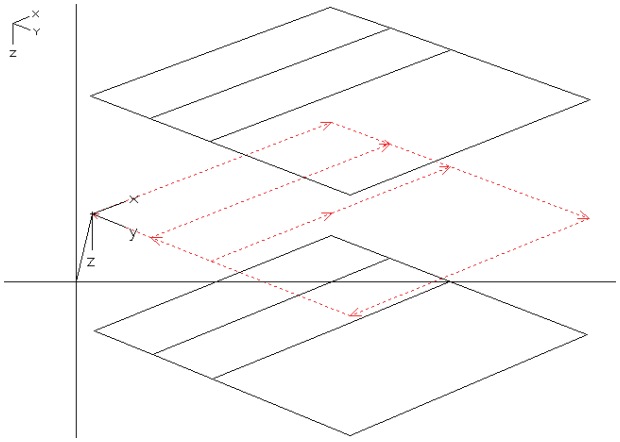
Now switch to the *3D* view and create the other levels using the copy function.

To do so, select the edges by dragging a box over all the objects. The selected objects will now be displayed with dotted lines.
Select *Copy directly* in the *Edit* menu and specify the displacement vectors:

1. Point of displacement vector: [0 0 0]
2. Point of displacement vector: [0 0 3]
2. Point of displacement vector: [0 0 -3]

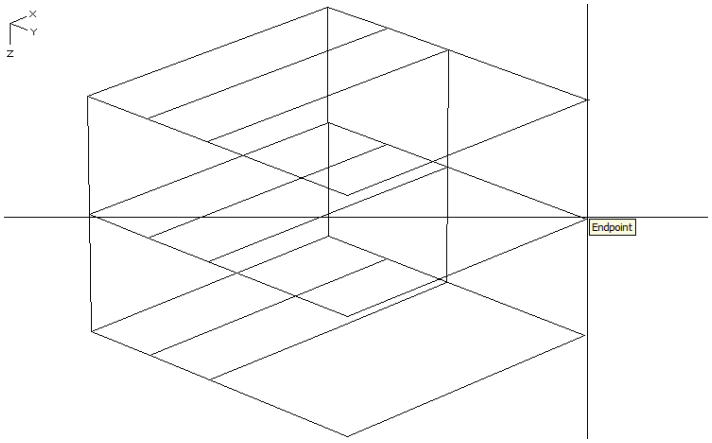
For 3D editing, always enter 3 coordinates in the order x, y and z. Based on the above coordinates, the displacement vectors point +/- 3 meters in z direction.

If you want to temporarily change the angle of view, click *Rotate with mouse* and move the cursor while holding down the left mouse button. Click the button again to stop the rotation.



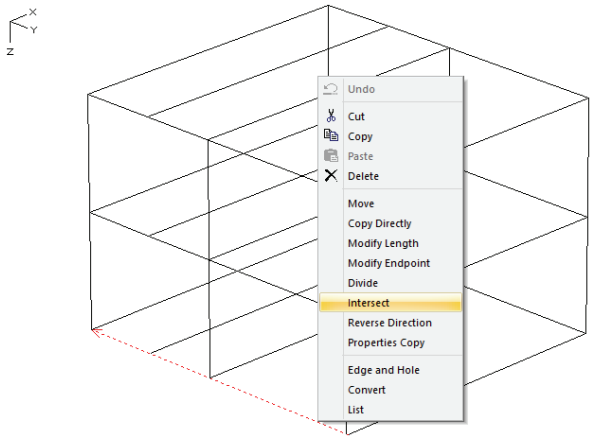
You can adjust the view to fit the new object dimensions using the *Zoom all* button.

Enter the missing edges between the levels. You can reference existing points and thus avoid having to enter additional coordinates manually.

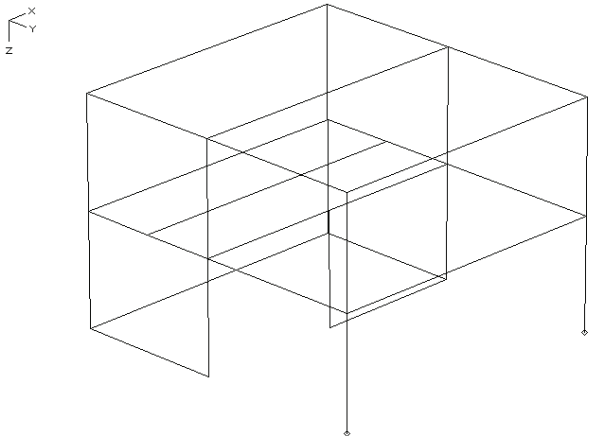


Select the unnecessary edges and *delete* them.

To shorten the lower edges select these edges with the right mouse button and choose the option *Intersect.* For border object select the relevant edge.



© InfoGraph GmbH, July 2015

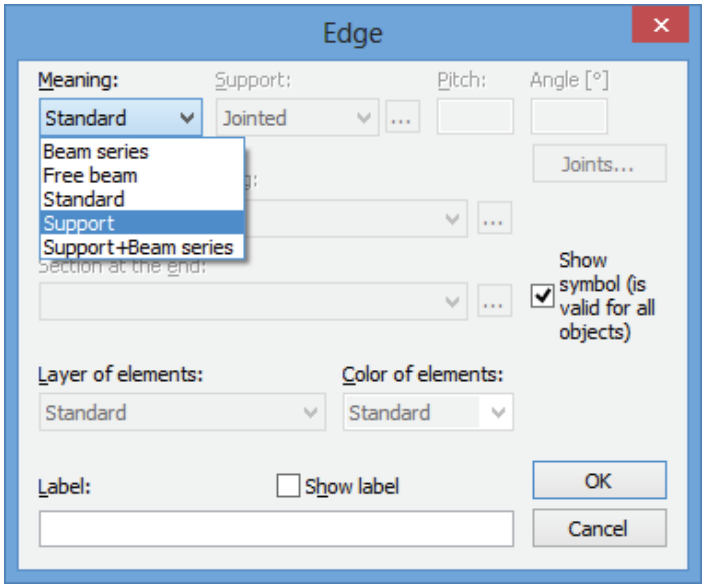


In order to create supports at the columns later we use the model object *Column* with the option *Point* at the foot points of the columns.

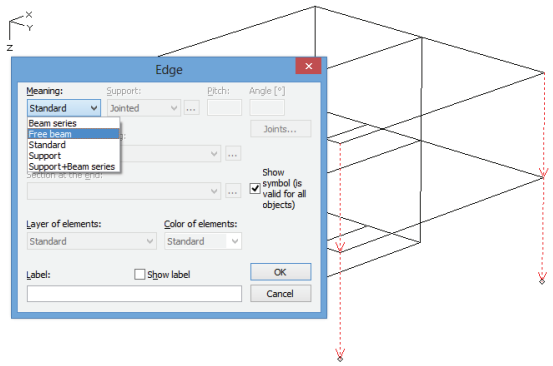
The wire frame model is completed.

Assigning Properties

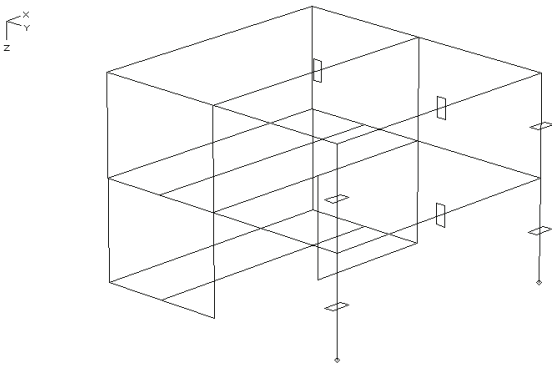
Additional properties can be assigned to the edges to be taken into account while generating the mesh. Double-click one or more edges to define the properties and set the wanted *meaning:*



Meaning	Result from the mesh generator
Standard	Area limitation or fixed line inside
Support	Line support on the edge
Beam series	Beam elements connected to the FE mesh, e.g. downstand beams
Free beam	Beam elements with independent intersection, e.g. columns or parts of frames
Support +Beam series	Line support and beam series

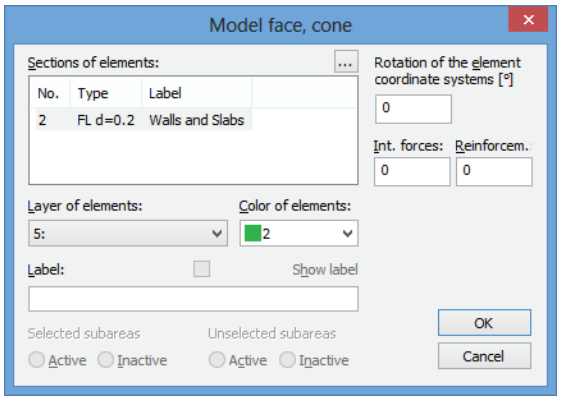


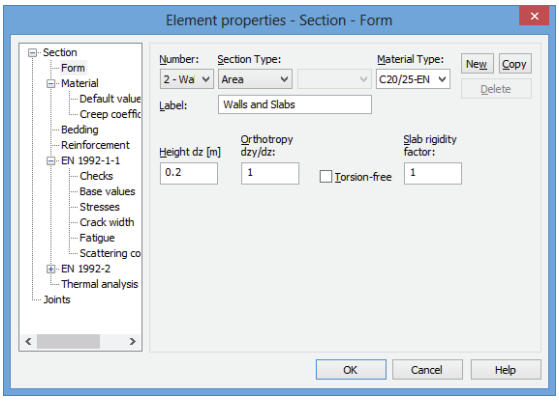
For the columns set *Free beam* with *pitch* = 2. Start the section dialog and define the polygon section with the function '...'.



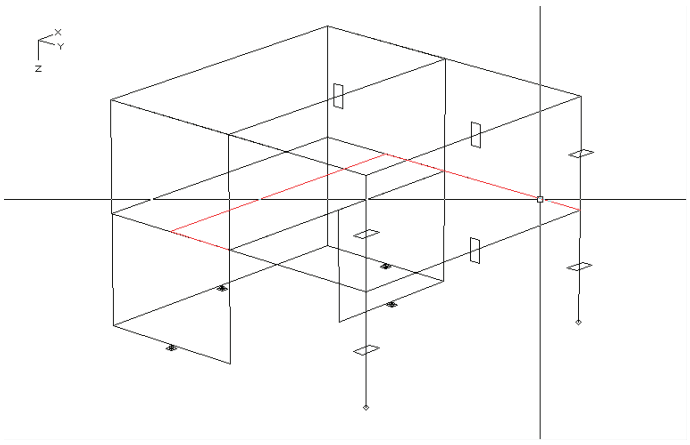
Assign the property *Support* to the supported edges and *Beam series* with the polygon section to the outer downstand beams.

Click the *Face* button and select the *Layer* and *Color* you want to use for the element mesh before defining the area (*Properties* option). Double-click inside the section window to open the section dialog. Set the properties for the area section and leave the dialog.

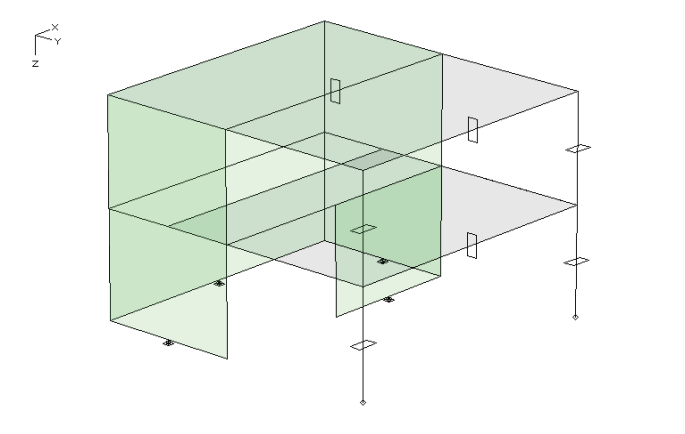




Now select the limiting objects for the first area. An area will be created as soon as it is fully defined by the selection. Change the *color* now and again if necessary.



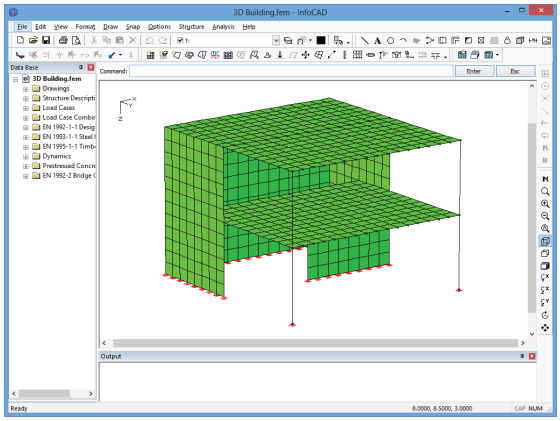
© InfoGraph GmbH, July 2015

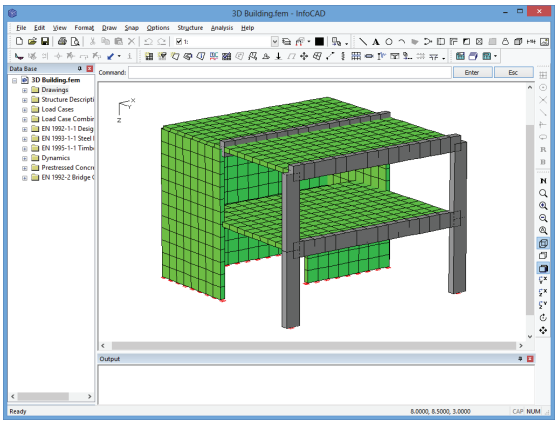


The geometric description of the system is now complete. Save the file and enter a name for the drawing.

Generating an Element Mesh

The model structure that is created serves as the basis for FEM mesh generation.
Click the *Mesh generation* button, select the *Form-sensitive* option and accept the suggested mesh width to 0.5 [m].





Click the *View options* button to access various settings for how the system is displayed. In the resulting dialog, activate the *Area fill of the elements* option to improve the 3D quality of the structure.
Check the structure using the *section representation*.

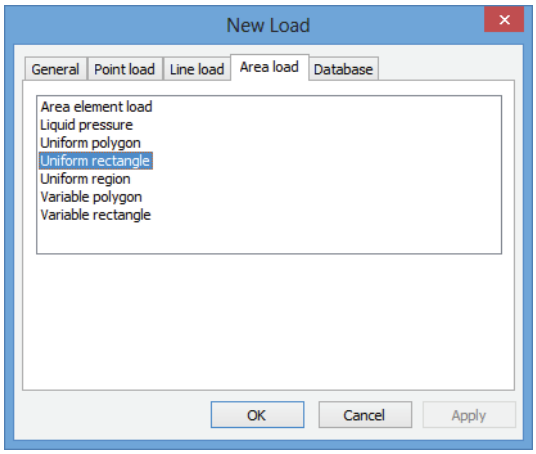
Defining Load Cases

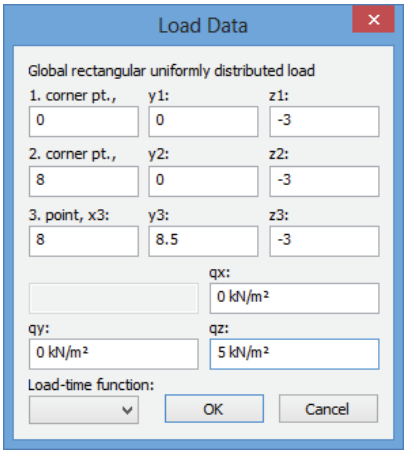
Open the *Load cases* folder in the database and double-click New load case.
A new load case will be created automatically and the load input and editing buttons will appear in the dialog bar.

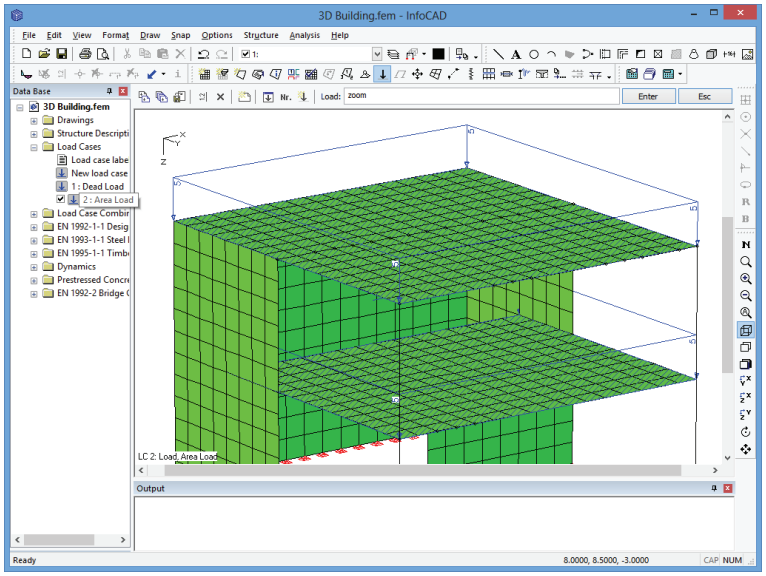
Nr.

For load case 1, select dead load with weighting 1 in the z direction, confirm your selection and then enter a load case label using the *Change Number* button.

Click New load case in the database again to create the new load case 2. Select the area load *Uniform rectangle* and define two rectangle area loads on the floors. When entering coordinates refer to existing nodes and use the functions *Zoom* and *Rotate with mouse*.

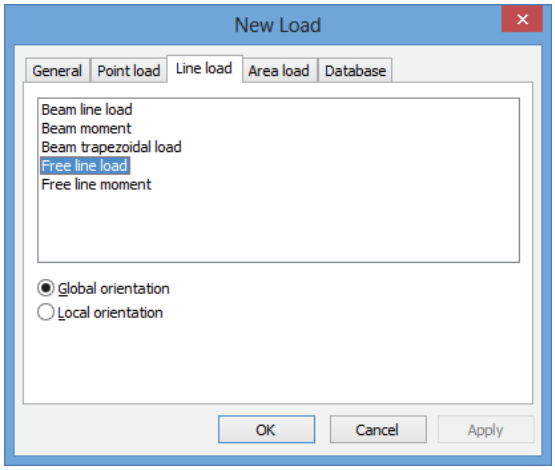


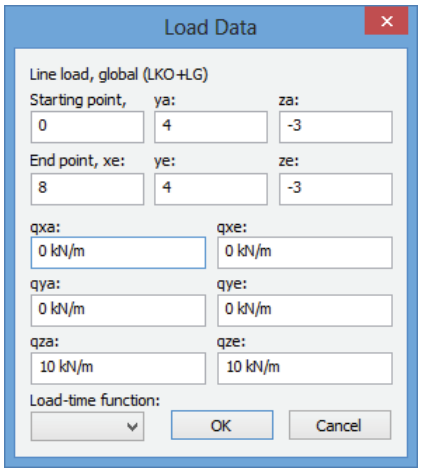


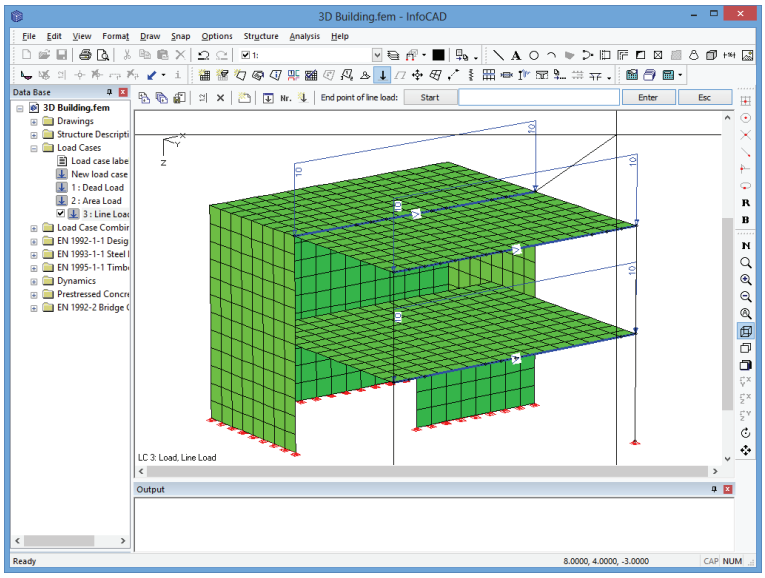


© InfoGraph GmbH, July 2015

For load case 3, select *Line load* for area elements with global orientation.

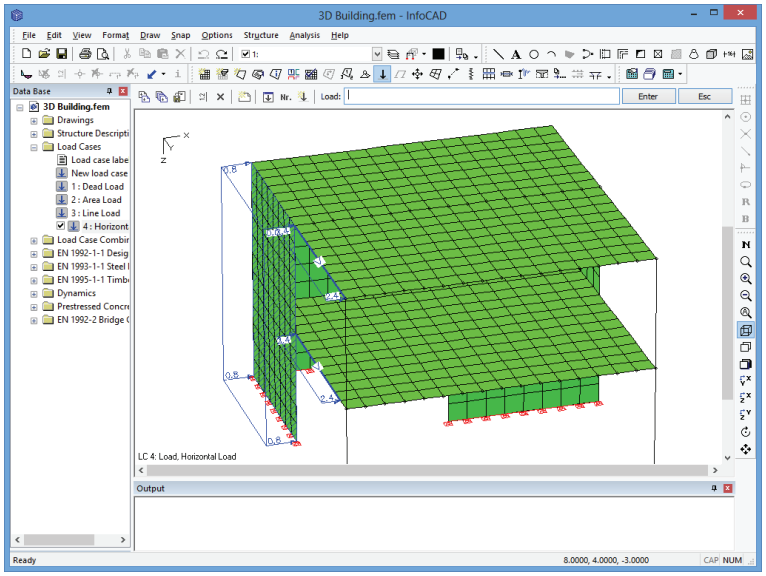






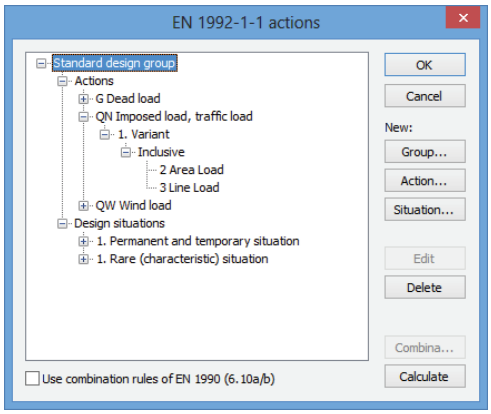
With the *Load representation* option you can scale the load ordinates.

For the fourth load case create two Line loads with 2.4 kN/m and one area load with 0.8 kN/m² in global x direction.



Load input is now complete and you can exit the *Load* dialog.

Describing Actions



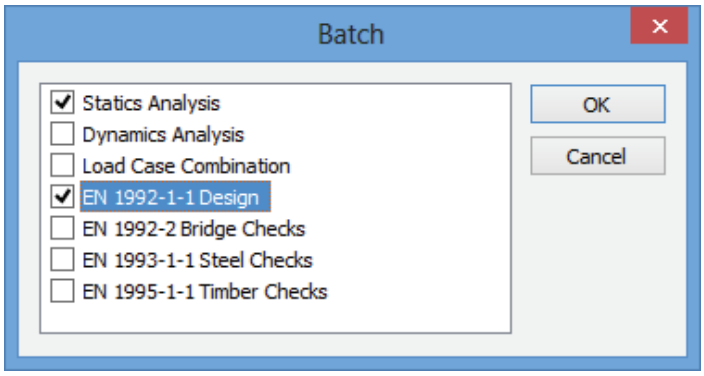
Open *Actions* in the *EN 1992-1-1 Design* folder of the database, assign the following actions to the load cases and create a *Permanent and temporary situation* and a *Rare (characteristic) Situation.*

G Dead load:	Load case 1 (dead load)
QN Imposed load, traffic load:	Load case 2 and 3 (area load, line load)
QW Wind load:	Load case 4 (horizontal load)

© InfoGraph GmbH, July 2015

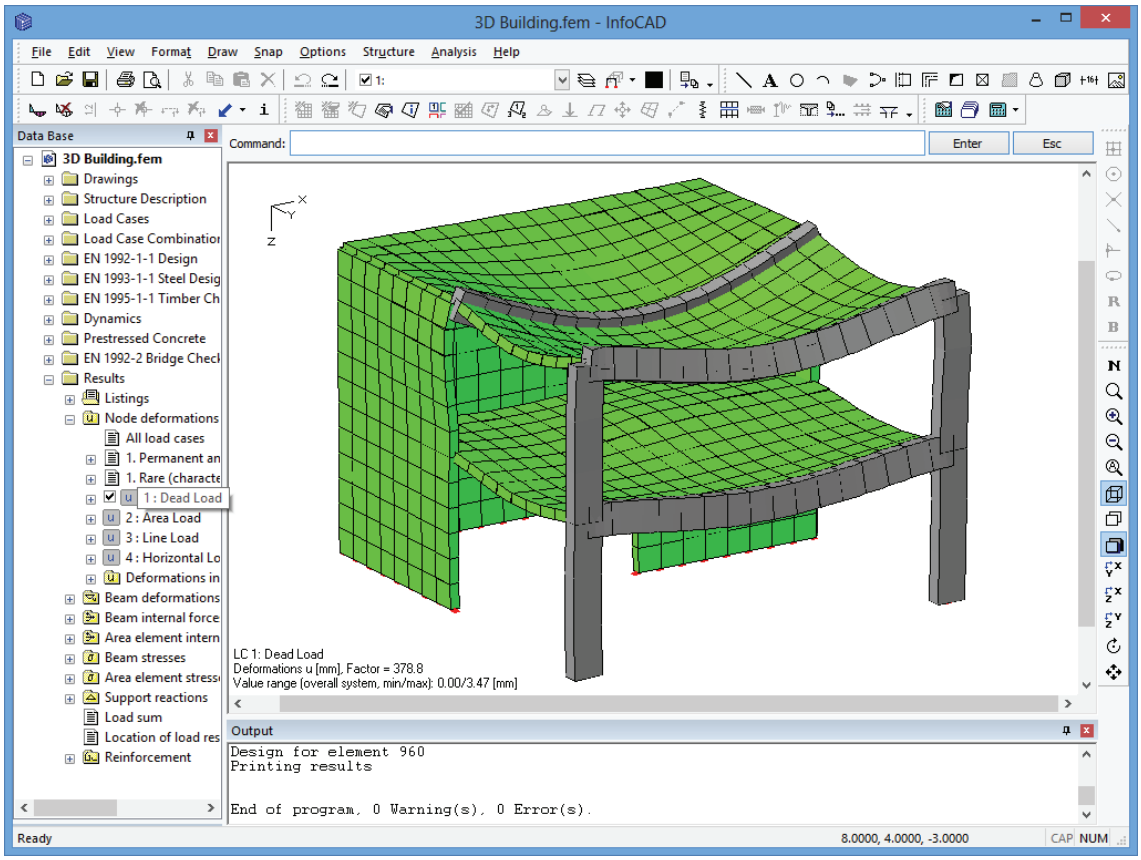
Calculation and Results

Begin the batch calculation with the options selected as shown below.

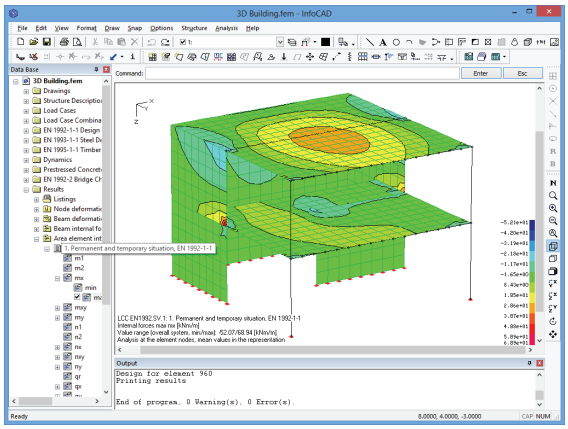


The selected calculations are performed automatically in consecutive order. You can follow the progress of the calculations in the output bar.

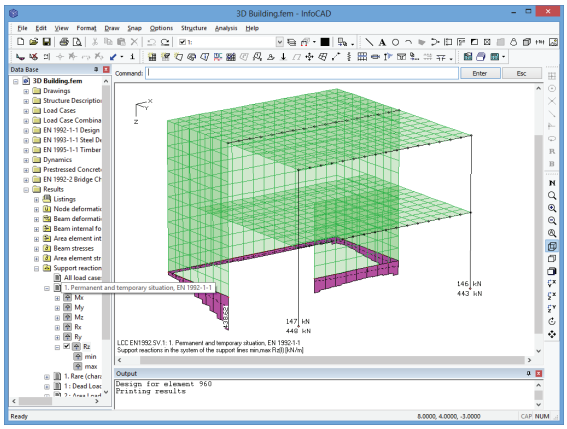
Once the calculation is finished, all the results are available in the database. Open the results category you need and select the appropriate view.



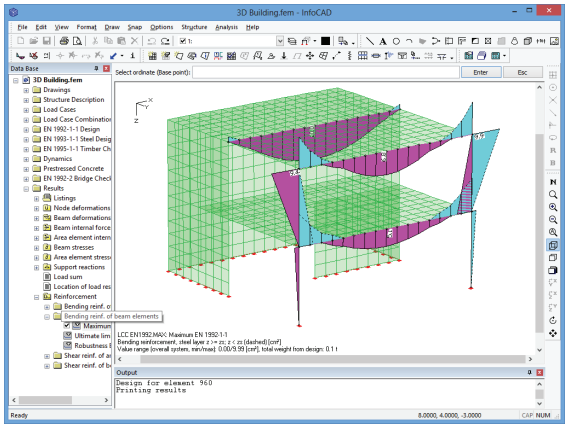
Deformations load case 1



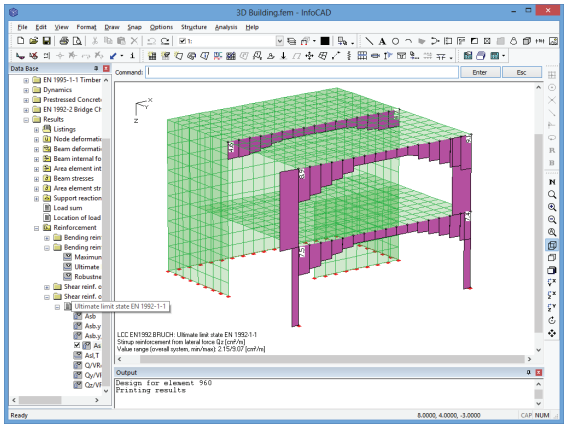
Colored progression of the moments mx



Support reactions



Longitudinal reinforcement of the beams

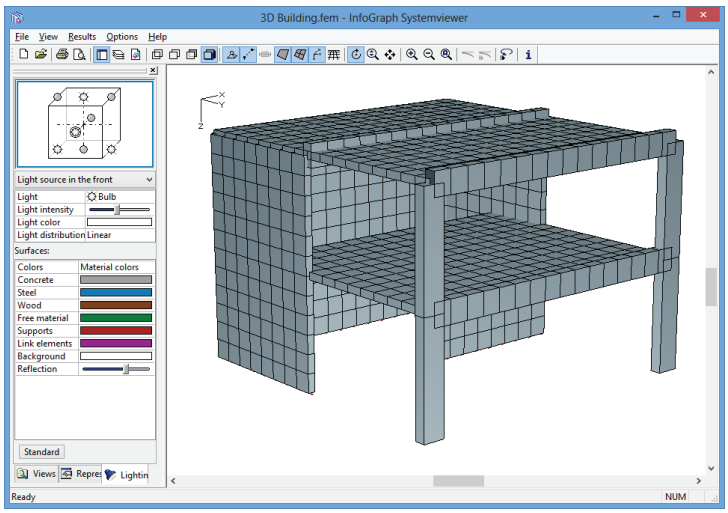


Lateral reinforcement of the beams and columns

InfoGraph Systemviewer

The InfoGraph Systemviewer allows you to visualize FEM and framework projects as well as to animate deformations from static or dynamic calculations.

Launch the Systemviewer from the *Programs / InfoGraph* folder in the *Start* menu and open the file you want to view. A variety of settings options are available to help you optimize the graphical output.



© InfoGraph GmbH, July 2015

Example 4: Prestressed Roof Girder

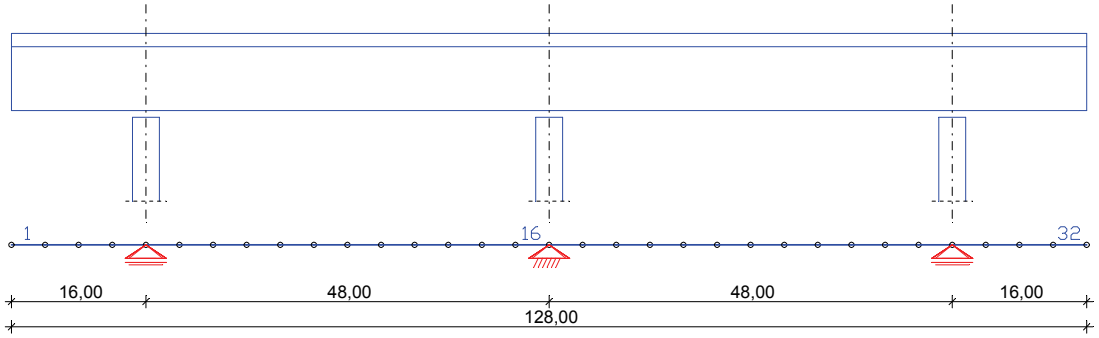
This example involves a wide-spanned roof construction that is represented as a continuous girder over two spans with a double-sided cantilever. To edit this system, perform the following work steps:

- Enter a static system
- Enter a tendon group
- Enter the loads
- EN 1992-1-1 Design
- Display the results

The structure is constructed for exposition class XC1. In accordance with EN 1992-1-1, Table 7.1N no check of decompression is necessary for this exposition class.

It is assumed that you have already gone through the previous examples. Hence, some of the processing steps will not be detailed in this example.

Task

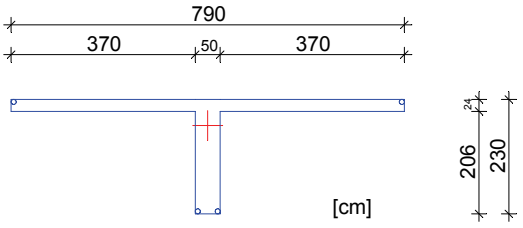


Static system and dimensions [m]

Material

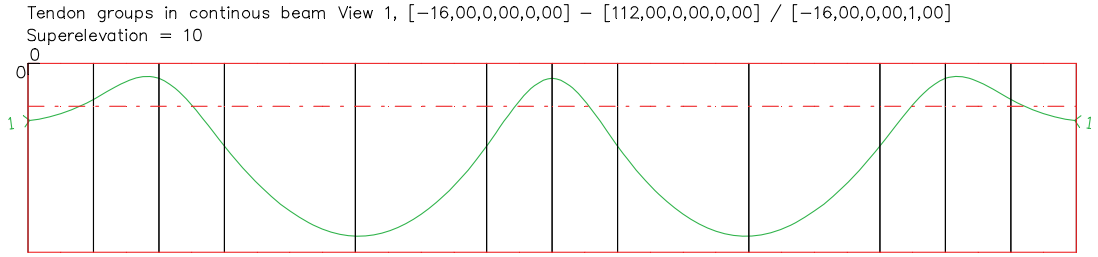
Concrete: C45/55-EN
Reinforcing steel: BSt 500/550, axis distance from edge 5 cm

Section



Tendon groups

Four bundled tendons are arranged in this example. The tendon group guide is shown in the next figure. The depicted tendon group ordinates z_V of the spline points refer to the upper edge of the section.



Tendon group ordinates zv [cm] at the base points

xv	0,00	8,00	16,00	24,00	40,00	56,00	64,00	72,00	88,00	104,00	112,00	120,00	128,00
1	70,0	44,0	18,5	101,0	210,5	101,0	18,5	101,0	210,5	101,0	18,5	44,0	70,0

Prestressing steel and prestressing system

Prestressing steel quality	St 1500/1770
Certification of the prestressing system	EC 2
Number of tendons in the bundle	4
Section surface area A_z	1800 mm²
E-modulus of the prestressing steel	195,000 MN/m²
0.1% strain limit (yield strength) of the prestressing steel $f_{p0.1k}$	1500 MN/m²
Tensile strength of the prestressing steel f_{pk}	1770 MN/m²
Permitted prestressing force of the tendon P_{m0}	2295 kN
Friction coefficients when increasing or releasing strain	0.2
Unintentional angle displacement of a tendon ß'	0.3 °/m
Slippage at prestressed tie bolt	6 mm
Duct diameter	82 mm
Allowance value for ensuring an over-stressing reserve κ	2

Scattering coefficients of the internal prestressing

Construction stage (r_{sup}/r_{inf})	1.1 / 0.9
Final state (r_{sup}/r_{inf})	1.1 / 0.9

Loads

Load case 1: Dead load (G1)
Load case 2: Additional loads q=11.06 kN/m (G2)
Load case 3: Traffic load (snow load) q=7.90 kN/m (Q)
Load case 10: Prestressing (P)
Load case 15: Creep-generating continuous load: G1+P+G2
Load case 20: Creep and shrinkage (CSR)

Coefficients: $\varphi_{t\infty} = 2.55$; $\rho = 0.8$; $\varepsilon_{cs,t\infty} = -24.8 \cdot 10^{-5}$

Creep-generating continuous load case: 15
The redistribution of internal forces between concrete and prestressing steel are taken into account.

© InfoGraph GmbH, July 2015

Static System

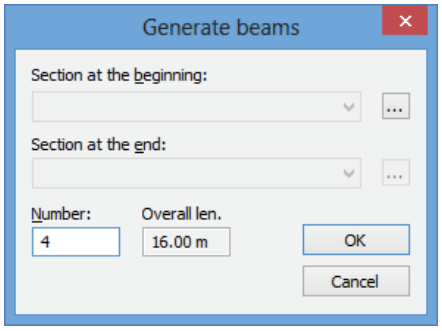
Launch the program or start a new project by clicking *New* and then selecting ***Finite Elements*** from the *Structure* menu.

To accurately measure the prestressing and the creep and shrinkage behavior, first divide the structure into beams with a length of 4 meters. Specify the supports based on a fork support of the system.

First switch to the xz view.

Click the button *Element creation* and select the element type *RS Beam*. For automatic division activate the option *Generate*. A crosshair will appear in the representation area and you will be prompted to enter the coordinates of the starting point in the dialog bar. Enter [0 0] to position the starting point at the origin. After you press [↵] or *Enter*, the end of generation line will be queried.

To enter additional coordinates, you should activate relative coordinate input by clicking the *Relative* button. A small axis system will now always appear at the last point inserted.
All successive coordinates you enter always refer to this point. Enter [16 0] to define the end of the generation line in x direction.



Enter 4 for the number of beams which shall be generated.

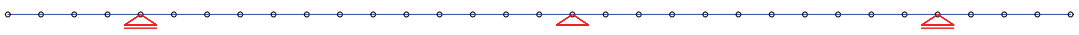
To temporarily enlarge the view, click *Zoom-* or *Zoom all*.

Now enter continuously the remaining section coordinates and choose the number of beams to be generated:

[48 0] - 12 Beams; [48 0] - 12 Beams; [16 0] - 4 Beams

Finish the beam input with [↵] or *Enter*.

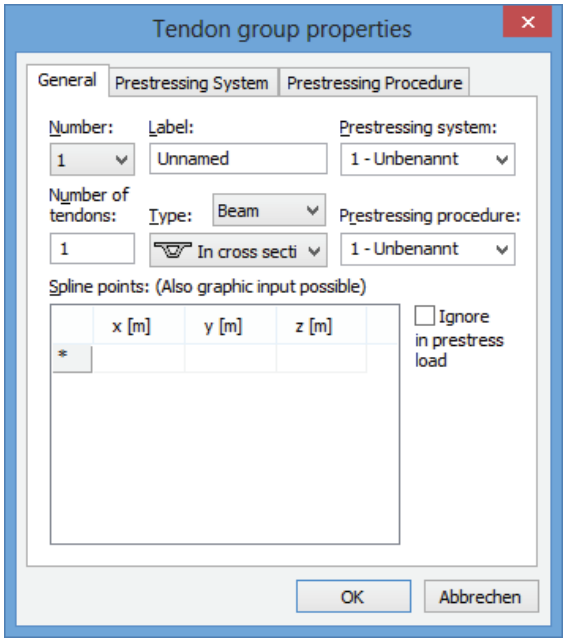
Now click the *Supports* button and choose the *Point support* option. You will be prompted to select the support nodes in the dialog bar. You can click the nodes directly with the crosshair.
During the support input take care to support the system free of restraint in the longitudinal direction and to define a fork support.



Entering a Tendon Group

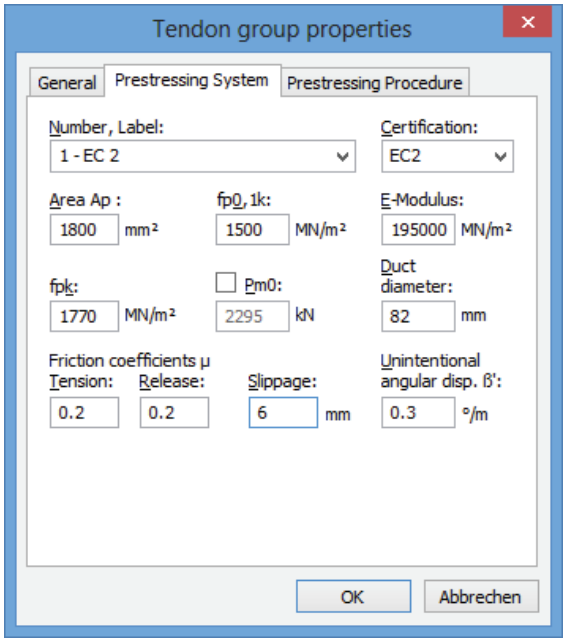
The definition of the tendon groups does not depend on the elements. The assignment and force introduction onto the structure occur while the *Prestressing* load case is being calculated. To avoid any conflicts, a distinction is made between beam and area/solid prestressing.
The tendon curve on which the calculation approach is based is represented by a 3D cubical spline function. The spline function is the curve that runs through all specified spline points with the least amount of curvature. The basis for the prestressing analysis is provided by a tendon group force curve that allows for strain increase, release and slippage. To include prestressing in the FEM analysis, you need to define a load case with the *Prestress* load type.

Use the *Prestressing* function to enter and edit tendon groups.



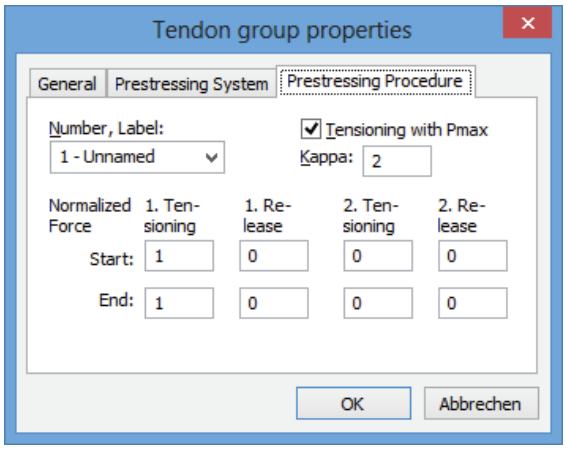
Click *Define* and then select *Beam* for the tendon group type in the resulting dialog.

Normally, the spline points are defined graphically.



The *Prestressing System* tab contains all the properties that are assigned to the tendon groups. They apply to each individual tendon of the group.

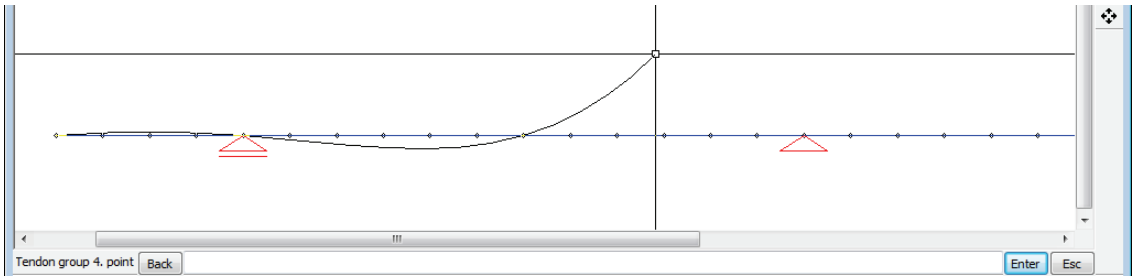
© InfoGraph GmbH, July 2015



If the allowance value κ is used, then the factor for the first over-stressing refers to the maximum permitted force P_{max}.

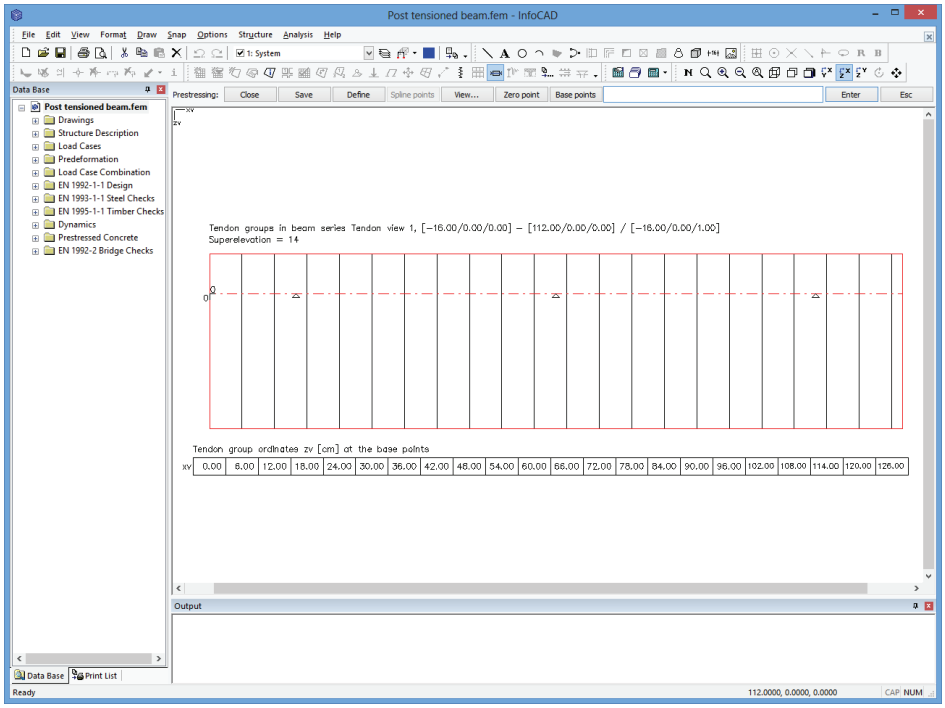
Using the factor specified for the release, the maximum prestressing force remaining in the tendon group is defined with respect to P_{m0}. A factor of '0' means no release.

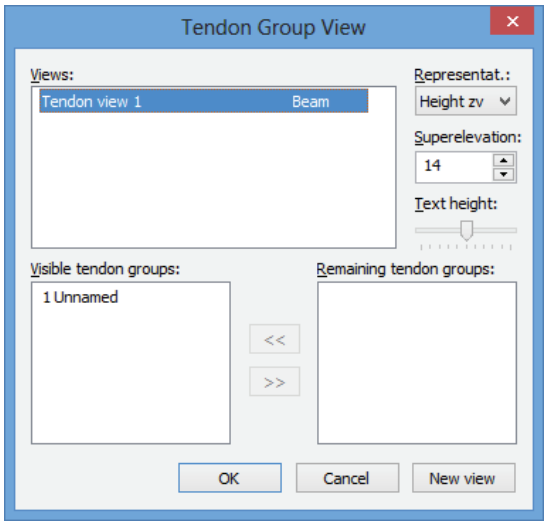
Exit the dialog and enter the spline points of the tendon group (start, supports, midspans) at the nodes of the beam array.



When you are done, click *OK*. The tendon group no. 1 is now positioned along the centroidal axis of the beam array.

Additional editing should take place in a section view. To access a section view, select the entire beam series with the *View/Beam* option and define the view plane from the first node to the last node in the global z direction.

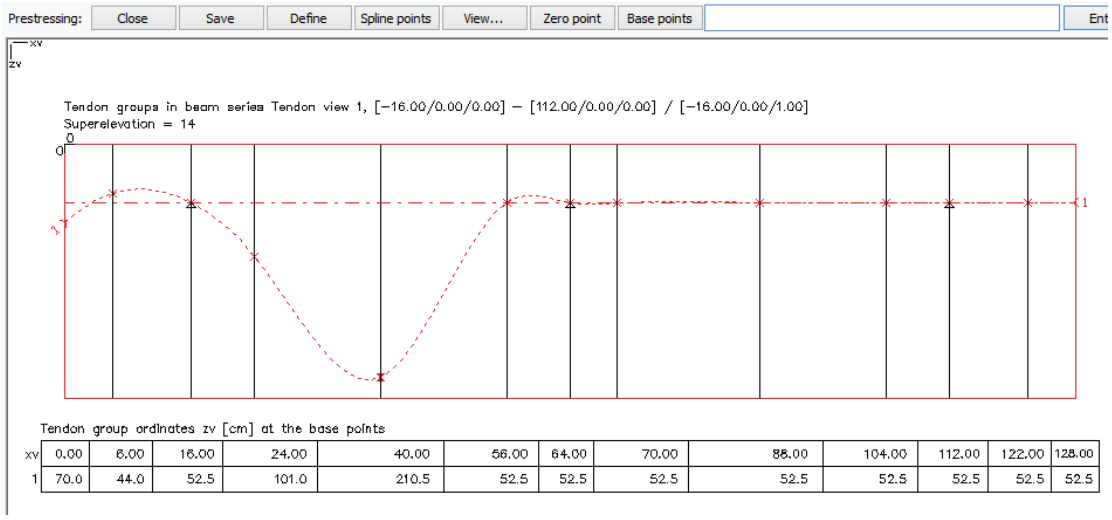

Post tensioned beam.fem - InfoCAD
Tendon groups in beam series Tendon view 1, [-16.00/0.00/0.00] - [112.00/0.00/0.00] / [-16.00/0.00/1.00]
Superelevation = 14
Tendon group ordinates zv [cm] at the base points
Output



Click the *View* option again and add tendon group 1 to the *Visible tendon groups* list.

The tendon group is now displayed in the selected section. Next, use the *Zero point* option to move the position of the reference system from the centroid to the upper edge of the section and then in the *Base points* option select *Spline points*.

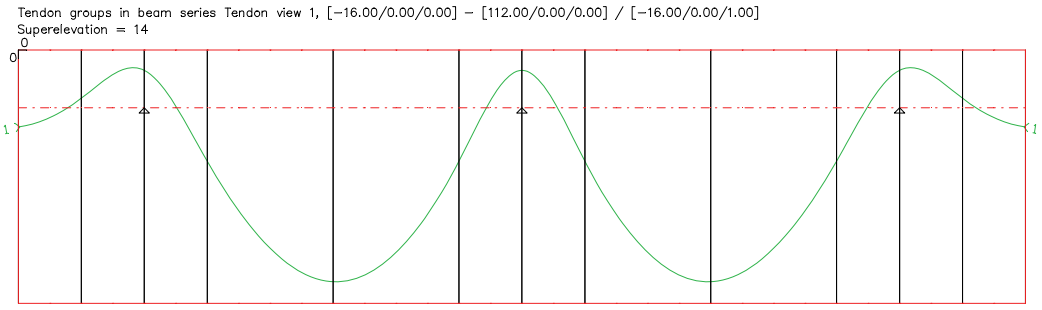
If the tendon group is selected, you can reposition it by first clicking the existing spline point and then the *Spline points* option. Only change the respective Zv position. The *Insert* option allows you to add additional spline points before the active spline point.



Tendon group ordinates zv [cm] at the base points

xv	0.00	8.00	16.00	24.00	40.00	56.00	64.00	70.00	88.00	104.00	112.00	122.00	128.00
1	70.0	44.0	52.5	101.0	210.5	52.5	52.5	52.5	52.5	52.5	52.5	52.5	52.5

Use this method to assign the desired geometry to the tendon group.



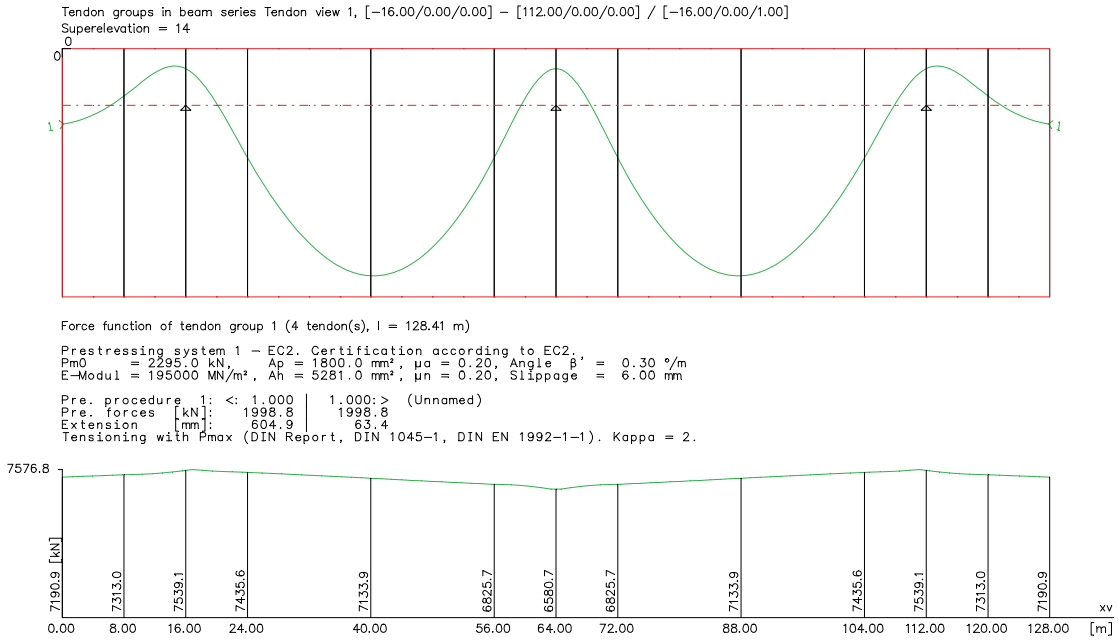
Tendon group ordinates zv [cm] at the base points

xv	0.00	8.00	16.00	24.00	40.00	56.00	64.00	72.00	88.00	104.00	112.00	120.00	128.00
1	70.0	44.0	18.5	101.0	210.5	101.0	18.5	101.0	210.5	101.0	18.5	44.0	70.0

In the tendon group view, the coordinates always refer to the reference system.

© InfoGraph GmbH, July 2015

Click the *Representation* option to activate the resulting prestressing force curve in the tendon group view:



Loads and actions according to EN 1992-1-1

Enter the following loads:

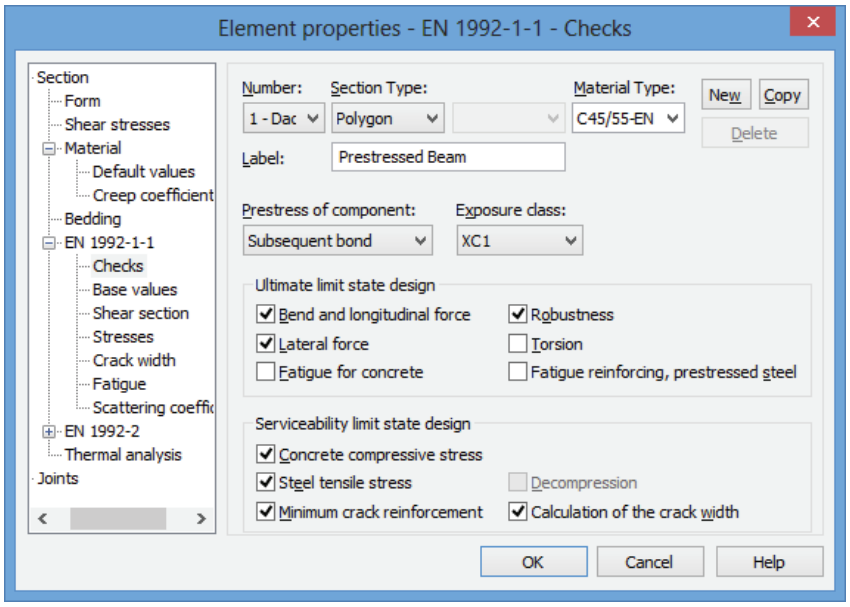
Load case 1: Dead load
Load case 2: Additional loads q = 11.06 kN/m
Load case 3: Snow load p = 7.90 kN/m
Load case 10: Prestressing
Load case 15: Creep-generating continuous load case.
Load cases 1, 2 and 10 are grouped into this load case with the *Insert* load type.
Load case 20: Creep and shrinkage.
The *Creep and Shrinkage* load type lets you calculate the redistribution of internal forces between the concrete and prestressed steel. Specify load case 15 as a creep-generating continuous load case.

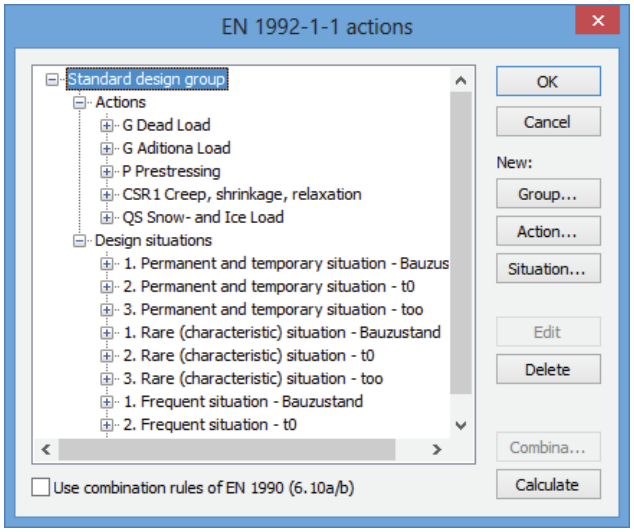
The following checks are carried out as a part of this example:

Checks at the ultimate limit state
- Minimum reinforcement for securing ductile member behavior
- Bending with or without longitudinal force or longitudinal force only
- Lateral force under consideration of the minimum level of reinforcement

Checks at the serviceability limit state
- Limiting the concrete compressive stresses
- Limiting the reinforcing steel stresses
- Limiting the prestressing steel stresses
- Minimum reinforcement for the crack width limitation
- Limiting the crack width via direct calculation

The following input is required for these checks:
- Select and adjust the settings for the checks in the element properties
- Define the actions

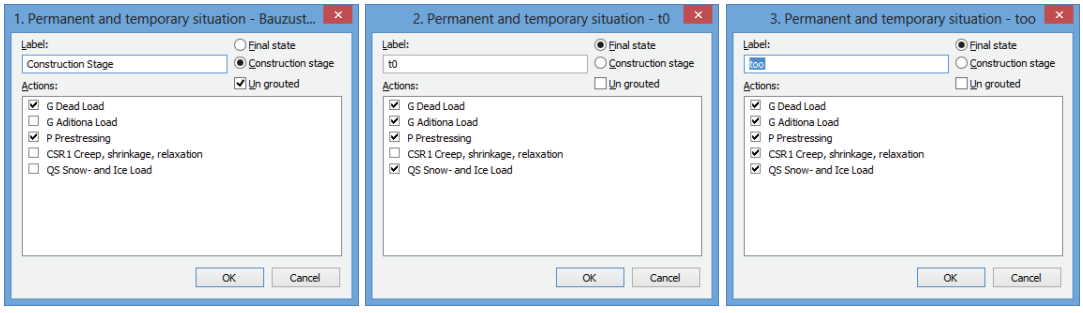




To carry out the desired checks the following situations are necessary:
- *Permanent and temporary*
- *Rare (characteristic)*
- *Frequent*
- *Quasi-continuous*

To take different (construction) stages into account, three situations are defined:

Construction stage:	Dead load G and prestressing P (tendon not grouted)
t_0:	Dead load G, P, additional load and snow load (tendon grouted)
t_∞:	Dead load G, P, additional load, snow load and CSR (tendon grouted)



© InfoGraph GmbH, July 2015

Calculation and results

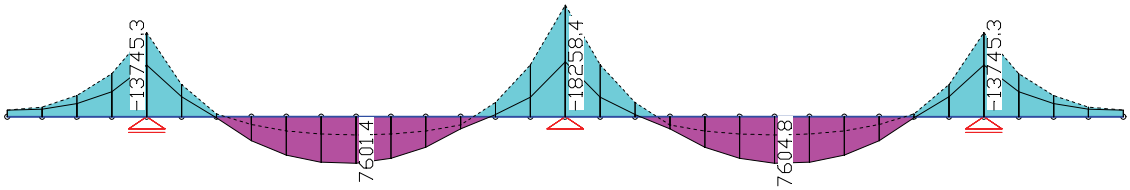
Perform the following calculations after all system specifications have been made:

- Statics
- EN 1992-1-1 Design

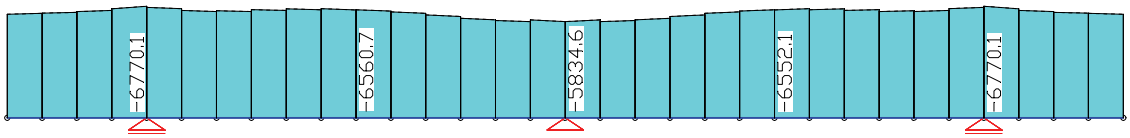
The necessary design situations are used for all checks depending on the requirement class. Each situation is checked independently and the maximum reinforcement for each steel layer is stored.

A number of different calculation results are shown below:

Internal forces

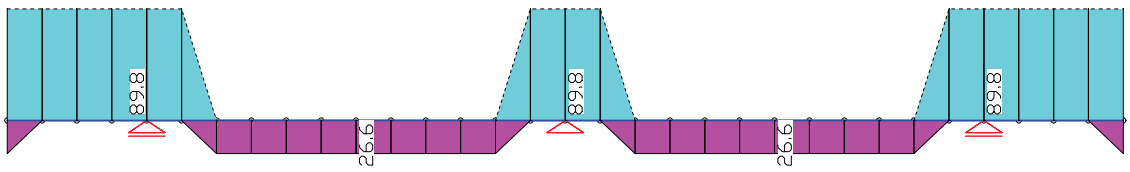


LCC EN1992.SV.3: 3. Permanent and temporary situation, EN 1992−1−1, too
Internal forces min,max My. 10953.67 [kNm] =
Value range (overall system, min/max): −18258.40/7604.76 [kNm]

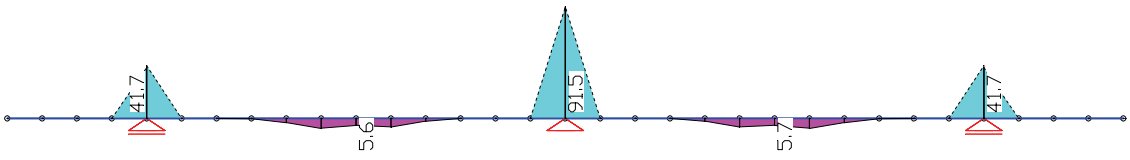


LCC EN1992.SV.3: 3. Permanent and temporary situation, EN 1992−1−1, too
Internal forces min,max Nx. 4061.57 [kN] =
Value range (overall system, min/max): −6770.14/−5834.61 [kN]

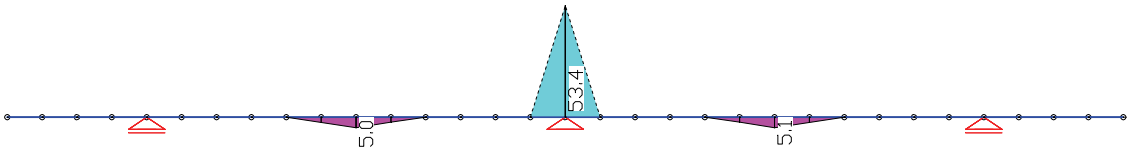
Longitudinal Reinforcement



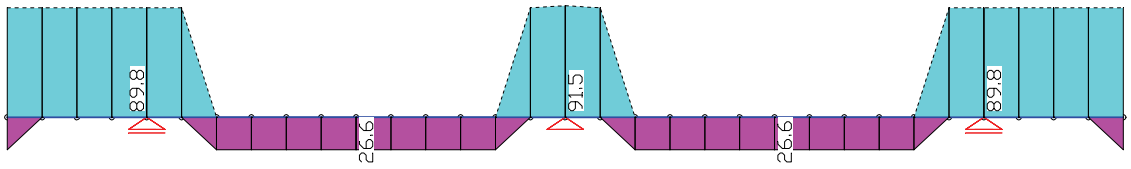
LCC EN1992.MIN: Robustness EN 1992−1−1
Bending reinforcement, steel layer z >= zs; z < zs (dashed). 53.89 [cm²] =
Value range (overall system, min/max): 0.00/89.82 [cm²], total weight from design: 5.5 t



LCC EN1992.RISS: Crack width limitation EN 1992−1−1
Bending reinforcement, steel layer z >= zs; z < zs (dashed). 54.89 [cm²] =
Value range (overall system, min/max): 0.00/91.49 [cm²], total weight from design: 0.7 t

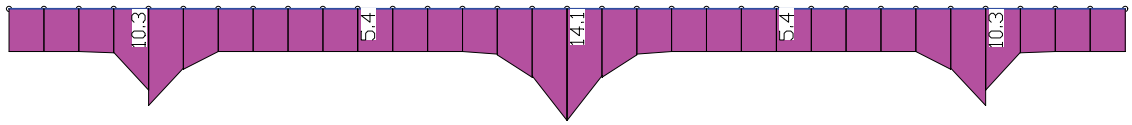


LCC EN1992.BRUCH: Ultimate limit state EN 1992–1–1
Bending reinforcement, steel layer z >= zs; z < zs (dashed). 32.06 [cm²] =
Value range (overall system, min/max): 0.00/53.44 [cm²], total weight from design: 0.2 t



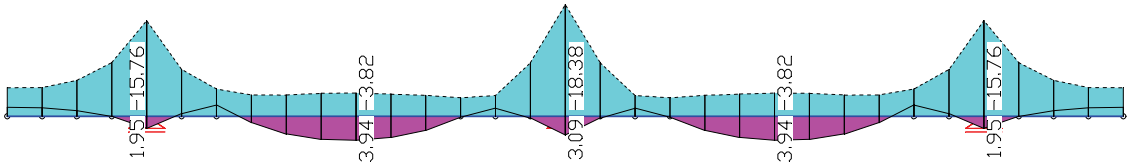
LCC EN1992.MAX: Maximum EN 1992–1–1
Bending reinforcement, steel layer z >= zs; z < zs (dashed). 54.89 [cm²] =
Value range (overall system, min/max): 0.00/91.49 [cm²], total weight from design: 5.5 t

Shear reinforcement



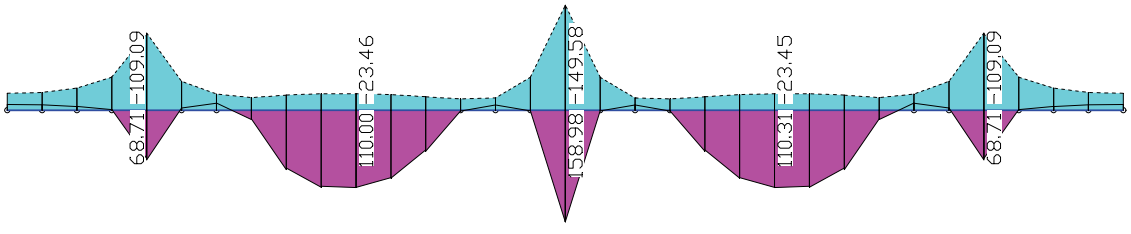
LCC EN1992.BRUCH: Ultimate limit state EN 1992–1–1
Stirrup reinforcement from lateral force Qz. 8.44 [cm²/m] =
Value range (overall system, min/max): 5.37/14.06 [cm²/m]

Concrete compressive stresses



LCC EN1992.C: Maximum, Rare (characteristic) situation, EN 1992–1–1
Concrete compressive stresses min,max Sigma.x. 11.03 [MN/m²] =
Value range (overall system, min/max): −18.38/3.94 [MN/m²]

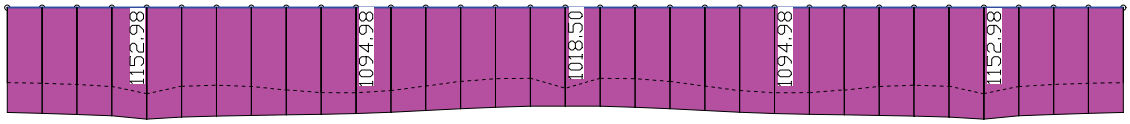
Reinforcing steel stresses



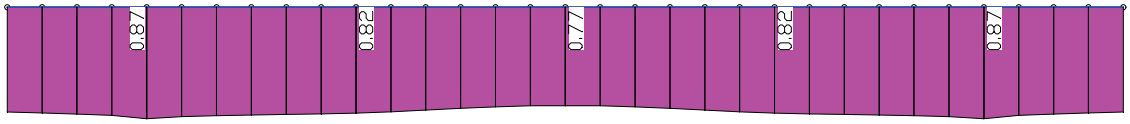
LCC EN1992.C: Maximum, Rare (characteristic) situation, EN 1992–1–1
Reinforcing steel stresses min,max Sigma.s. 95.38 [MN/m²] =
Value range (overall system, min/max): −149.58/158.98 [MN/m²]

© InfoGraph GmbH, July 2015

Prestressing steel stresses

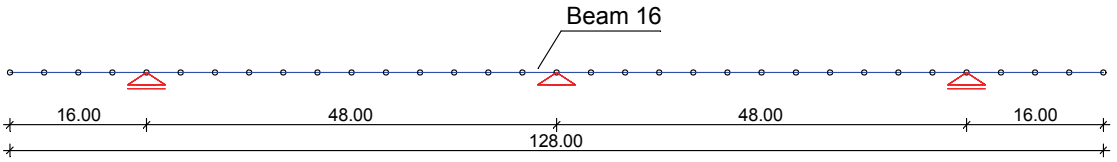


LCC EN1992.C: Maximum, Rare (characteristic) situation, EN 1992-1-1
Prestressing steel stresses min,max Sigma.p. 691.71 [MN/m²] =
Value range (overall system, min/max): 729.55/1153.00 [MN/m²]



LCC EN1992.C: Maximum, Rare (characteristic) situation, EN 1992-1-1
Prestressing steel stresses Sigma.p/Sigma.p,perm. 0.52 [-] =
Value range (overall system, min/max): 0.77/0.87 [-]

Excerpt from the 'EN 1992-1-1 Design' log (short version)



Design of longitudinal reinforcement

```
(M) Nominal reinf. for robustness as per EN 1992-2, 6.1 (109) (Charact. C.)
(R) Nominal/requ. reinforcement as per 7.3.2/4 for crack width limitation
    Increase of reinforcement due to crack width check is marked by "!".
    Ap'     Part of prestr. steel area Xi1*Ap which was used to reduce req.As
    Xi1     Bond coefficient for prestressing steel as per Eq. (7.5)
(B) Design of reinforement at ultimate limit state
    In case of dominant bending, compression reinforcement is marked with "*"
    For section areas acc. to 6.1.(5) the conrecte strain is not limited
    The minimum reinforcement acc. to 9.2.1.1 and 9.3.1.1 is not determined
    For compressive members the minimum reinf. acc. to 9.5.2 is considered
```

Beam No.	Se.	Lo.	Reinforcement Lay.	Type	Nx [kN]	My [kNm]	Mz [kNm]	Ap' [cm²]	req.As [cm²]
16	1	1	1	M	-85,63	-4604,92	0,00	.	44,91
				R	0,00	0,00	0,00	.	0,00
				B	-5946,80	-8500,50	0,00	.	0,00
			2	M	-85,63	-4604,92	0,00	.	44,91
				R	0,00	0,00	0,00	.	0,00
				B	-5946,80	-8500,50	0,00	.	0,00
			3	M	-1,32	-1558,17	0,00	.	0,00
				R	0,00	0,00	0,00	.	0,00
				B	-6698,11	-1120,99	0,00	.	0,00
			4	M	-1,32	-1558,17	0,00	.	0,00
				R	0,00	0,00	0,00	.	0,00
				B	-6698,11	-1120,99	0,00	.	0,00
16	1	2	1	M	127,26	-12798,56	0,00	.	44,91
				R	-5251,14	-10872,84	0,00	.	45,75!
				B	-5834,61	-18258,39	0,00	.	26,72
			2	M	127,26	-12798,56	0,00	.	44,91
				R	-5251,14	-10872,84	0,00	.	45,75!
				B	-5834,61	-18258,39	0,00	.	26,72
			3	M	0,05	-7871,14	0,00	.	0,00
				R	0,00	0,00	0,00	.	0,00
				B	-6580,63	-5633,71	0,00	.	0,00
			4	M	0,05	-7871,14	0,00	.	0,00
				R	0,00	0,00	0,00	.	0,00
				B	-6580,63	-5633,71	0,00	.	0,00

Design of shear reinforcement

The percentage of nominal reinforcement acc. to Eq. (9.5N) is considered.

VRd, TRd Design value of maximum absorbable lateral force, torsional moment
Angle   Angle cot Theta between the compressive strut and the beam axis
Asb,Asl.T Req. stirrup reinf. from lateral force and torsion, torsional reinf.
Asl   Req. longitudinal reinf. acc. to Fig. 6.3 for req. Asb.

Beam No.	Loc.	Qy/ VRd	Asb.y Angle	Asb.y [cm²/m]	Qz/ VRd	Asb.z Angle	Asb.z [cm²/m]	Asl [cm²]	Q/VRd+ Mx/TRd	Asb.T [cm²/m]	Asl.T [cm²]
16	1	0,00	2,50	0,00	0,34	2,50	8,45	.	.	.	.
	2	0,00	2,50	0,00	0,56	2,50	14,06	.	.	.	.

Check of crack widths

The check is led by direct calculation of the crack width.
The final long. reinforcement as the maximum from robustness, crack and bending reinf. incl. a possible increase resulting from the fatigue check is decisive.

wk   Calculated value of crack width as per 7.3.4 [mm]
wmax   Permissible crack width as per specification [mm]
Sigma.c Maximal concrete edge stress in state I [MN/m²]
         (CC) Charact. (rare), (TC) Frequent, (QC) Quasi-continuous combination

Beam No.	Se.	C.	Lo.	Reinf. Layer	Nx [kN]	My [kNm]	Mz [kNm]	Sigma.x [MN/m²]	wk [mm]	wmax [mm]
16	1	TC	1	.	-5352,12	-4473,47	0,00	0,04	-.-	0,20
			2	1	-5251,14	-10872,8	0,00	2,75	0,20	0,20

Check of concrete compressive stress

For the check, a cracked concrete section (II) is assumed if the tensile stress from the decisive c. exceeds the value of fctm. Otherwise, a non-cracked section (I) is used. If the strain is not absorbable on cracked section, (I*) is marked.

Sigma.x,min Total maximal longitudinal compressive stress [MN/m²]
Sigma.x,per = 0,60*fck for Charact. C. (CC) as per 7.2 (2)
(t,b)   Position of the edge point: above, below of centre

Beam No.	Se.	Loc.	Sigma.x,min [MN/m²]	Sigma.x,per [MN/m²]	Se.-Pnt.	Side t	Side b	Period	Situation
16	1	1 (I)	-8,85	-27,00	9	.	x	Final	CC.3
		2 (I)	-18,38	-27,00	9	.	x	Final	CC.3

Check of steel stress

For the check, a cracked concrete section is assumed.
For tendon groups without bond and/or for situations before grouting, the prestressing steel stress is checked acc. to Eq. (5.43).

Type S Long. reinf. from N and M, layer number, Charact. C. (CC)
Type P Prestressing steel, Tendon number, Charact. C. (CC)
Sigma.s,per = 0.80 * fyk resp. 1.0 * fyk (CK) as per 7.2 (5)
Sigma.p,per = 0.75 * fpk as per 7.2 (5)

Beam No.	Se.	Lo.	Steel Type	No.	As [cm²]	Sigma.s [MN/m²]	per. [MN/m²]	Situation
16	1	1	S	1	44,91	-2,12	400,00	CC.3
			S	2	44,91	-2,12	400,00	CC.3
			S	3	0,00	.	400,00	CC.1
			S	4	0,00	.	400,00	CC.1
			P	1	72,00	936,81	1275,00	CC.1
16	1	2	S	1	45,75	107,00	400,00	CC.3
			S	2	45,75	106,99	400,00	CC.3
			S	3	0,00	.	400,00	CC.1
			S	4	0,00	.	400,00	CC.1
			P	1	72,00	913,98	1275,00	CC.1

 © InfoGraph GmbH, July 2015

InfoGraph GmbH
Kackertstrasse 10
52072 Aachen, Germany
Tel. +49 - 241 - 889980
Fax +49 - 241 - 8899888
E-Mail: info@infograph.de
http://www.infograph.eu

InfoGraph

Software für die Tragwerksplanung